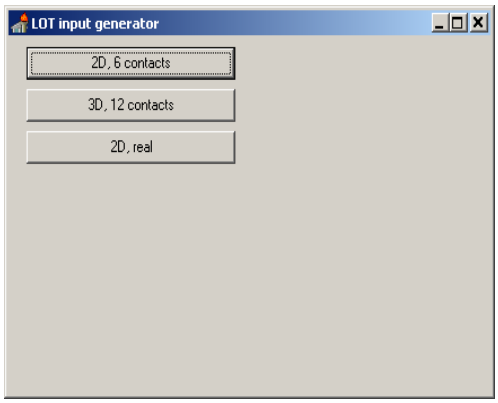
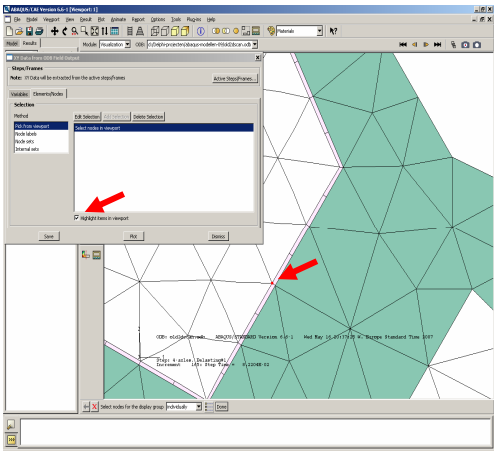
Report 7-07-170-5 Lifetime Optimisation Tool, LOT, User's Guide

December 2007

M. Huurman
L.T. Mo





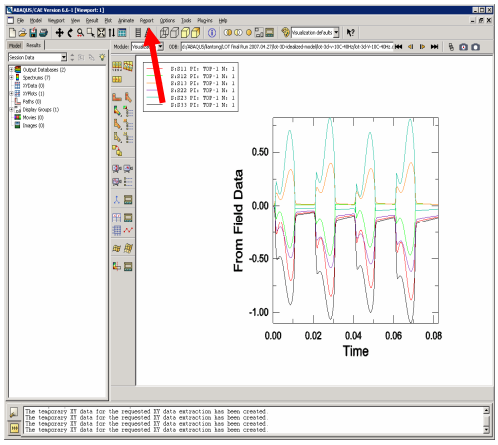


TABLE OF CONTENTS

1 INTRODUCTION 3

2 LOT INPUT GENERATOR 5
2.1 Introduction 5
2.2 Model selection 5
2.3 Geometrical input 7
2.3.1 Idealised models 7
2.3.2 2D scan/photo models 9
2.4 Materials input 9
2.5 Load input 10
2.6 ABAQUS input file generation 11

3 RUNNING LOT IN ABAQUS 13

4 POST PROCESSING ABAQUS LOT RUNS 21
4.1 Introduction 21
4.2 2D idealised model 23
4.2.1 Adhesive zone damage 23
4.2.2 Mortar damage 27
4.3 3D idealized model 31
4.3.1 Adhesive zone damage 31
4.3.2 Mortar damage 34
4.4 2D photo model 39
4.4.1 Adhesive zone damage 39
4.4.2 Mortar damage 43

1 INTRODUCTION

LOT, Lifetime Optimisation Tool, is a tool which focuses on the mechanical phenomena that take place inside a Porous Asphalt (PA) mixture during the passage of a tyre. LOT is developed by the Delft University of Technology and tries to explain ravelling - which is the main cause for PA reconstruction - on the basis of a meso scale mechanical analysis of the PA structure.
LOT was developed as part of a larger project. For background information of the tool and the hypotheses that lay behind it, reference is made to the following LOT reports:

Report 7-07-170-1 Lifetime Optimisation Tool, LOT, Main Report [1].
Report 7-07-170-2 Fatigue in mortar and adhesive zones; measurements, test interpretation and determination of model parameters [2].
Report 7-07-170-3 LOT, Mortar response; measurements, test interpretation and determination of model parameters [3].
Report 7-07-170-4 Lifetime Optimisation Tool, Sample preparation and laboratory testing for the LOT research program [4].

In this User's Guide emphasis is purely on the practical usage of LOT. No attention is paid to any theoretical issues.

LOT is a highly sophisticated tool. Users of the tool should have ample understanding of WINDOWS including standard multi-purpose tools such as Microsoft Excel. The kernel of LOT operates within the ABAQUS Finite Element (FE) platform. It is strongly advised that users of LOT have ample understanding of FE-modelling in general. With respect to mastering ABAQUS reference is made to the ABAQUS Version 6.6 User's Manuals [5].

This User's Guide aims to give LOT users a jump start. Usage of WINDOWS, Excel, and ABAQUS in general is far beyond the scope of this guide. It is anticipated that users of LOT are well able to integrate stress signals over time by application of Excel. Similarly it is anticipated that LOT users are able to determine dissipated energy from a given stress-strain signal. For this reason this guide explains how jobs can be run and how relevant data can be extracted from ABAQUS. The process that follows after data is exported to ASCII-file and is thus available for Excel or any other general purpose tool, is not discussed. This Guide concentrates on the following computer programs.

- LOT input generator
  The LOT input generator is a simple computer program written at the Delft University of Technology. The program allows quick and simple generation of complex input files to be fed into the ABAQUS FE-package.
- ABAQUS
  ABAQUS is a very powerful FE-package that is globally available on the market. ABAQUS was selected as the platform for LOT.

In chapter 2 the LOT input generator is discussed. It is explained that the use of this program makes the creation of complex ABAQUS input files an easy task. Hereafter, in chapter 3, it is explained how these input files are fed into ABAQUS. In chapter 4 it is explained how relevant ABAQUS output can be exported from ABAQUS to ASCII-file. These ASCII-files can then be imported into Excel or any other general purpose tool for further analyses.

2 LOT INPUT GENERATOR

2.1 Introduction

Place all LOT input generator files in a single folder, see figure 2.1. The LOT input generator is now ready for use.

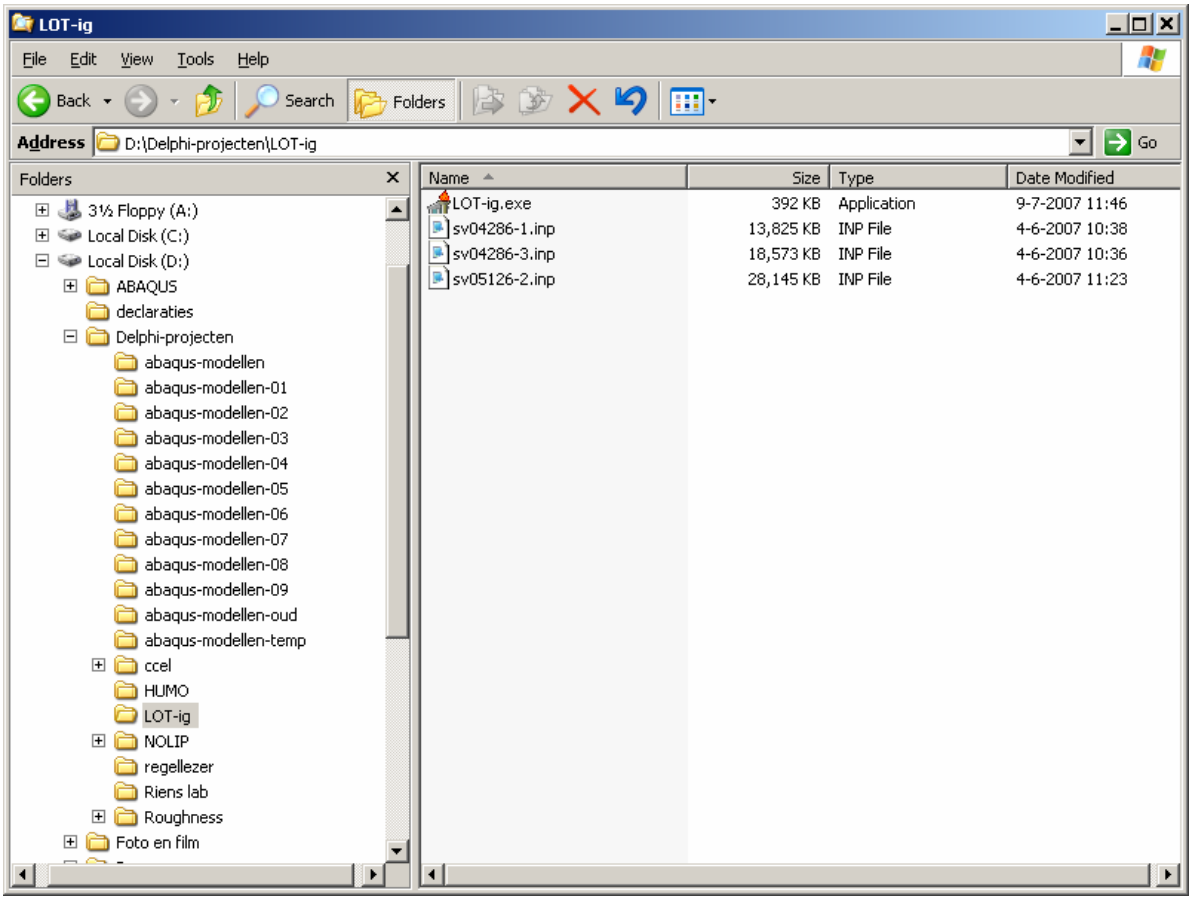


Figure 2.1 All LOT input generator files in a single folder.

2.2 Model selection

Start the LOT input generator by clicking ***LOT-ig.exe***. The following Window will appear.

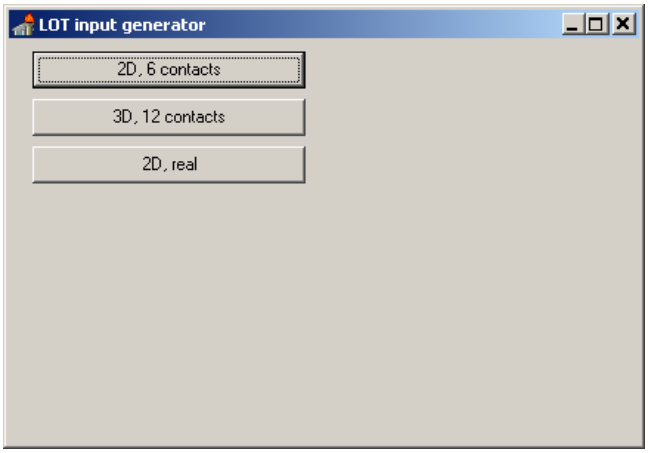


Figure 2.2 Main window of the LOT input generator.

Select the type of model that you want to use by clicking the relevant button. Impressions of the available models are given in the figures 2.3 to 2.5.

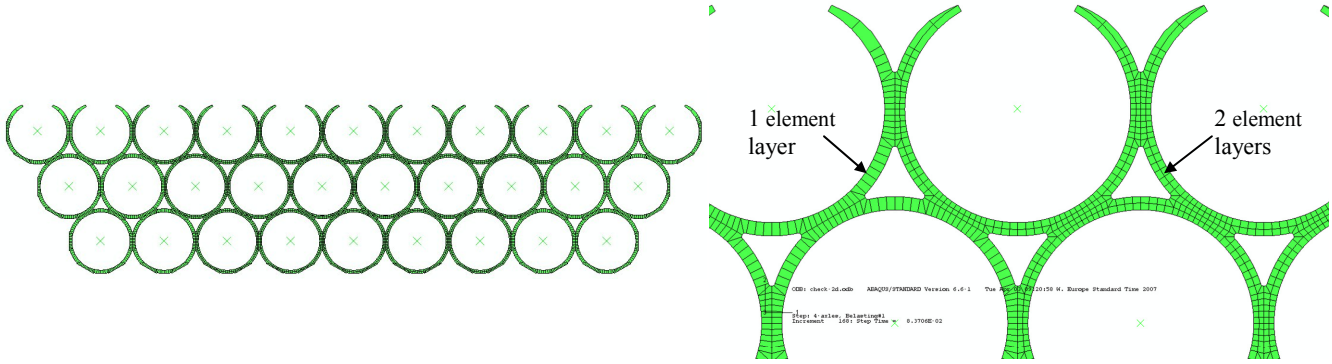


Figure 2.3 Idealised 2D, 6 contacts model (left) and model detail (right).

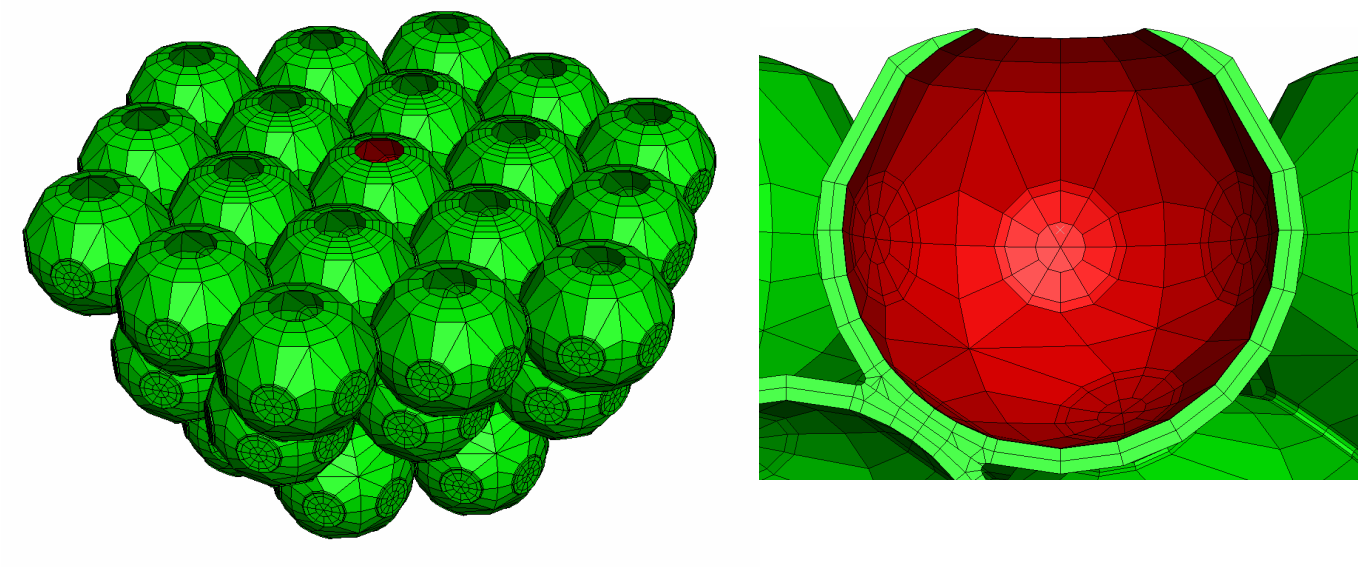


Figure 2.4 Idealised 3D, 12 contacts model (left) and model detail (right).

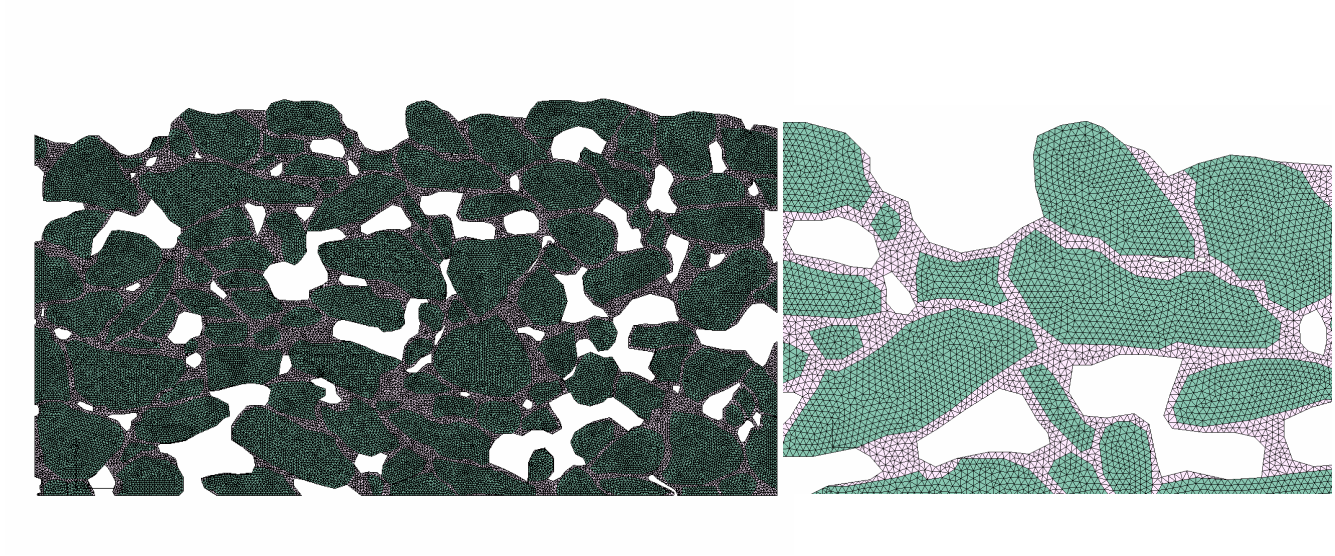


Figure 2.5 2D, real model (left) and model detail (right).

After selection of the desired model type, a window with four tabs will appear. These tabs have the following names. ***Geometry/Mesh***, ***Materials***, ***Load***, ***Make INP-file***. Behind the first tab mesh related inputs can be fed into the program. Similarly the second tab allows inputting material (component) related inputs, i.e. adhesive zone stiffness and mortar response behaviour. The third tab gives access to a window in which the loading can be specified. Finally, the fourth tab provides the means to generate the ABAQUS input file as per your inputs.

*NOTE: It is of vital importance that the tabs are utilised from left to right, each time closing the relevant window by clicking **OK**. When the various tabs are not opened and closed in the correct order the input file that is generated will most likely be faulty.*

2.3 Geometrical input

2.3.1 Idealised models

The geometry of the idealised models is fully determined by 7 parameters. These parameters are listed behind the ***Geometry/Mesh*** tab, see figure 2.6. The LOT input generator checks whether inputs are within realistic limits.

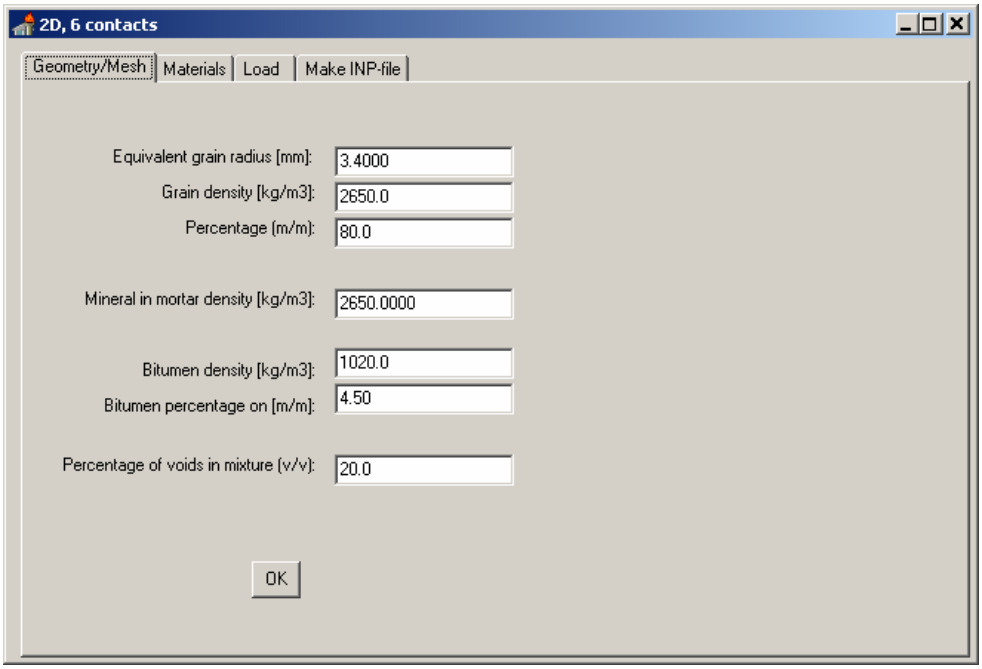


Figure 2.6 Mesh input window for the 2D, 6 contact and the 3D, 12 contact models.

First the equivalent grain size has to be inputted. This grain size is dependant on the mineral grading and may be determined by application of the following equation. Please note that the LOT input generator demands the equivalent grain radius as input while Equation 2.1 gives the equivalent grain diameter.

$$D = \sum_{i=1}^{n} D_i \cdot fr_i \Big/ \sum_{i=1}^{n} fr_i \qquad (2.1)$$

Where D: Equivalent grain diameter; n: number of fractions in the stone fraction, i.e. D>2mm [-]; i: fraction counter [-]; D_i: Diameter of stones in the i^{th} fraction [mm]; $fr_{i;}$: material quantity in the i^{th} fraction [m/m].

Secondly the specific density of the stone mineral needs to be fed into the program. Of course the size of the stone fraction (m/m) compared to the total mineral content (filler+sand+stone) needs to be specified.

Thereafter the specific density of the mineral in mortar, i.e. sand and filler, has to be inputted into the program. Similarly the density of the bitumen and the amount of bitumen in the mixture needs to be known.

It should be clear that the discussed parameters give the LOT input generator a good indication of mixture composition. Insight into the volumetric composition of the mixture is obtained after the last input parameter brings in the volume of voids. With this the complete structure of the idealised mixture is known and a mesh can be generated.

Close the window by clicking ***OK***.

2.3.2 2D scan/photo models

In the case of 2D scan/photo models the geometry is fully determined by the photo or scan used as a basis for the desired model. In this case the model geometry is completely known after selection of one of the available models. As is shown in figure 2.7 three models are available to date.

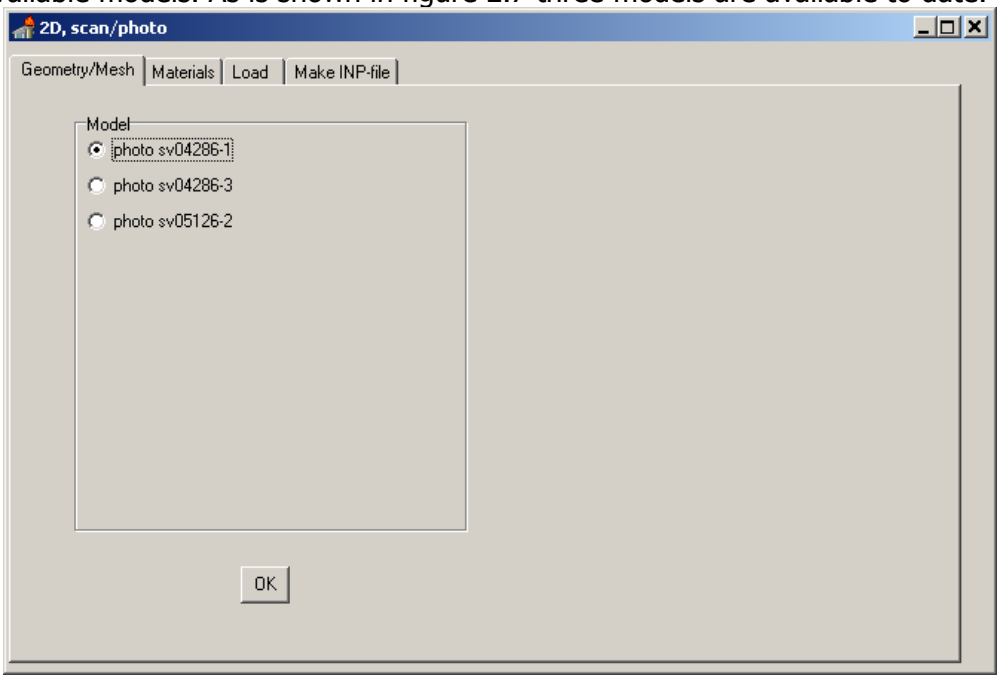


Figure 2.7 Mesh input window for the 2D scan/photo models.

Close the window by clicking ***OK***.

2.4 Materials input

Figure 2.8 gives the window for materials input. Since the definition of materials, or material components, does not depend on the selected type of model this window is the same for all models.

The window demands the response characteristics of the mortar and the adhesive zone.

The adhesive zone is modelled linear elastic. Its stiffness is defined as generated stress per mm deformation of the adhesive zone. It is assumed that deformation over the adhesive zone in reality remains limited in any case due to the limited thickness of the adhesive zone. In the models the adhesive zones have a thickness of 10 micrometer, i.e. 0.01 mm.
It is suggested to estimate the stiffness of the adhesive zone on the basis of the stiffness of the mortar. Since the thickness of the adhesive zone is limited to 0.01 mm the following equations apply.

$$k_n = \frac{E^*}{0.01\,mm} \tag{2.2}$$

$$k_s = \frac{G^*}{0.01\,mm} \tag{2.3}$$

Where: k_n : normal stiffness of the adhesive zone [MPa/mm], k_s : shear stiffness of the adhesive zone [MPa/mm], 0.01 mm: thickness of the adhesive zone.

For proper determination of mortar stiffness reference is made to [3].

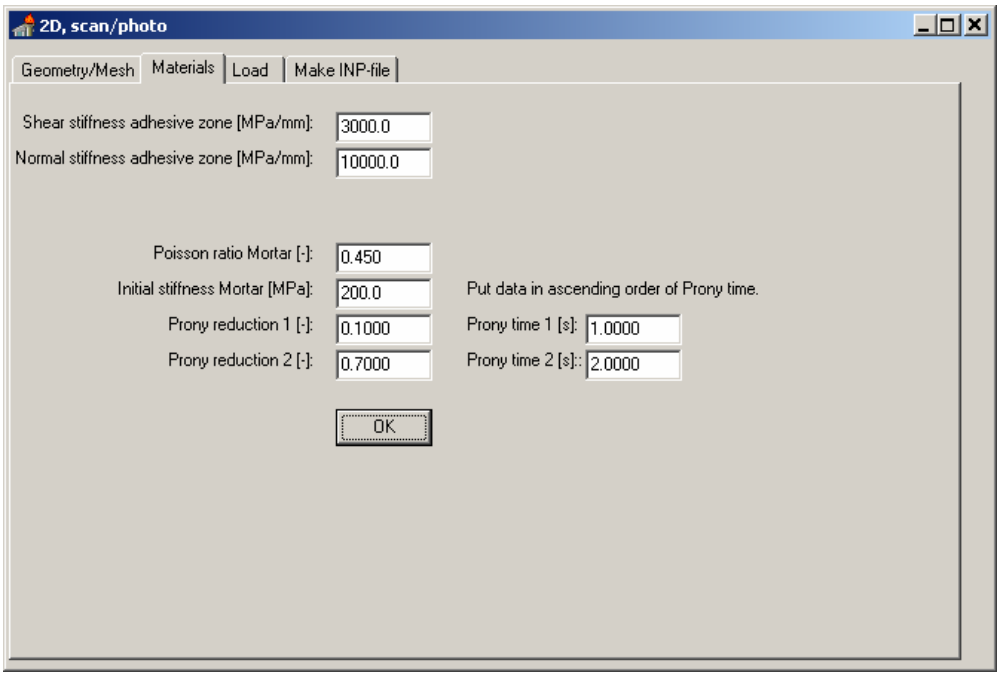


Figure 2.8 Material input window.

The response behaviour of the mastic is defined by Prony series visco-elasticity. This definition demands that the initial mortar stiffness is known. Secondly the reduction of stiffness due to creep effects is to be known. In this case these effects are defined by two reduction factors in combination with two parameters that describe the rate of relaxation (Prony time). It is important that the 1st Prony term has a shorter Prony time than the 2nd Prony term.
The Prony series model implemented in LOT is made 3D by effects dictated by the Poisson's ratio. It is suggested to always use a Poisson's ratio of 0.45. Again reference is made to [3] for proper determination of mortar response characteristics.

Close the window by clicking ***OK***.

2.5 Load input

Figure 2.9 gives the window for load input. The load signals applied to LOT are based on interpretation of measurements reported in literature. Basically the load signals applied to the model follow from fixed ratios between vertical and lateral and between vertical and longitudinal stress introduced by a tyre. The effects of engine output act on longitudinal stress and are computed on theoretical basis.

LOT is, at the moment, tuned to compute the PA response under four wheel loads, i.e. a free rolling wheel, an engine driven wheel, a second free rolling wheel and finally a second engine driven wheel.

Some parameters are required to fine tune the load signals. These parameters are.

- Axle load: No comment
- Length of contact area: The length of the tyre foot print
- Length of unloaded area: The length of the unloaded area between the free rolling and the engine driven wheel. It is strongly advised to not use the real length between the contact patches of two wheels. This would result in very long ABAQUS computations.

- Local contact pressure: The local contact pressure may well differ from the average contact pressure. Values up to twice the average pressure are realistic. The local vertical contact pressure governs all other stresses applied to the model.
- Time for a tyre to pass: This parameter effectively controls the speed of the moving tyre.
- Engine output: no comment.
- Throttle position: Percentage of the engine output that is put into the gearing system
- Loss in gearing: Engine output lost in the gearing system.
- Steps in load: Number of time steps during the time it takes a tyre to pass over a certain point. This parameter may affect the accuracy of output. It is advised to use a value of 20 or larger.

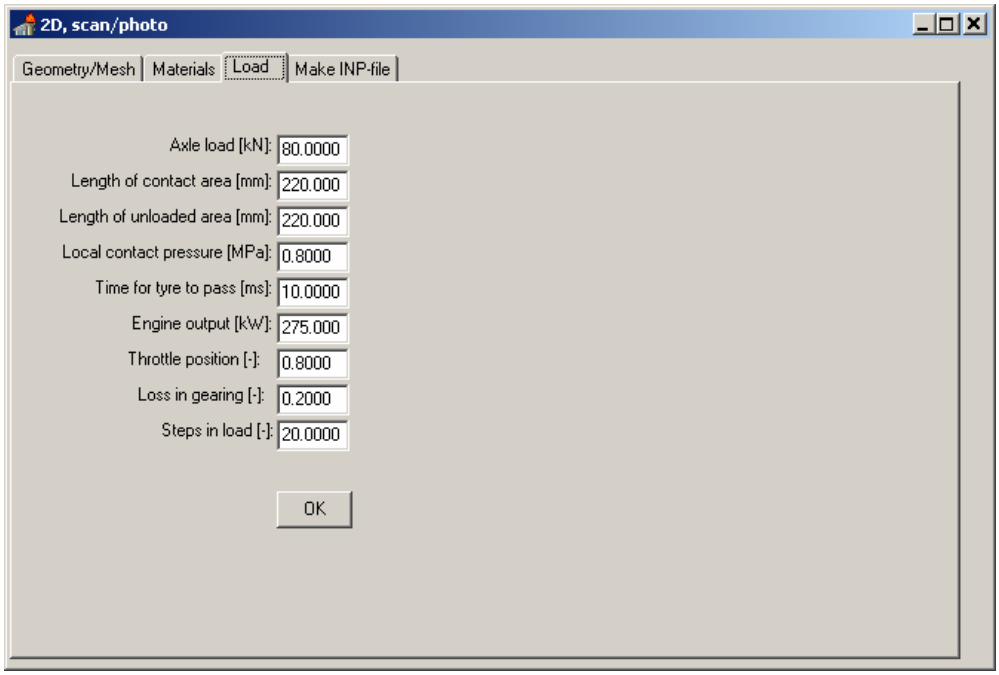


Figure 2.9 Load input window.

Figure 2.9 gives the window for load input. Since the definition of the model loading is not dependant on the selected type of model a similar screen is found behind the ***Load*** tab of all model types.

Close the window by clicking ***OK.***

2.6 ABAQUS input file generation

After the previously discussed tabs have been closed in the correct order by subsequent clicking ***OK*** the ABAQUS input file may be generated by the LOT input generator. Hereto the tab ***Make INP-file*** should be opened. The window shown in figure 2.10 will now appear. Two inputs are required. The first is the job name. This can be any text line that reflects the calculation that you are about to make. Secondly the name of the ABAQUS input file needs to be fed into the program. Pleas note that the file extension ***.inp*** is added automatically.

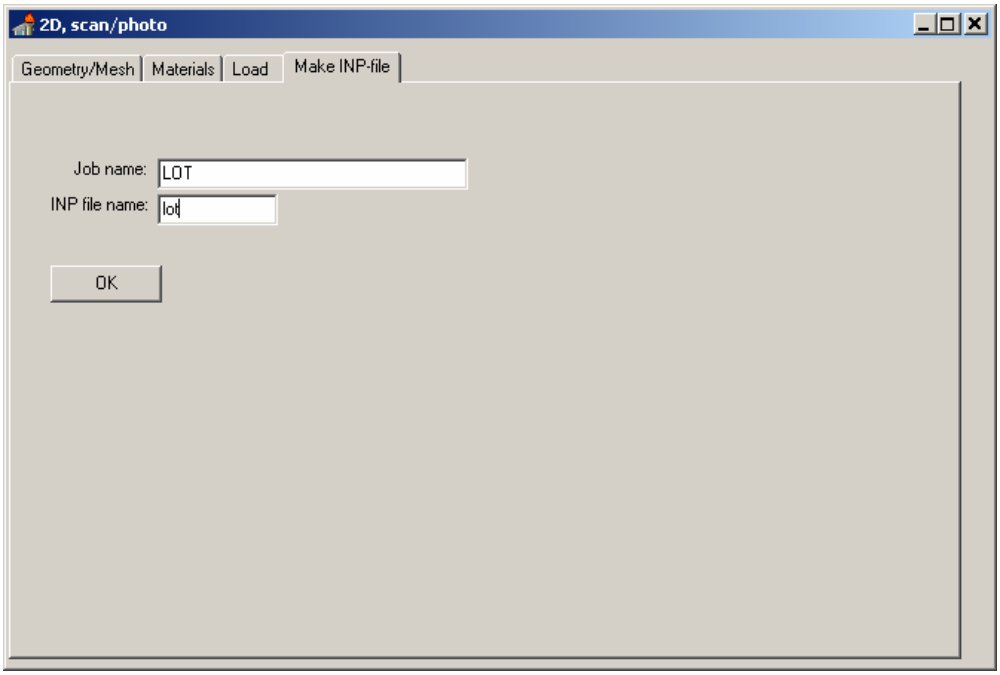


Figure 2.10 Window for job name input.

Close the window by clicking ***OK***.

It is stated that the ABAQUS inp-file that is generated by the LOT input generator is put in the folder which also contains the LOT input generator program itself. The user is advised to move the inp-file to any, more appropriate folder.

3 RUNNING LOT IN ABAQUS

Start ABAQUS CAE (Complete ABAQUS Environment). The following screen will appear. Close any window that may appear over the black viewport area.

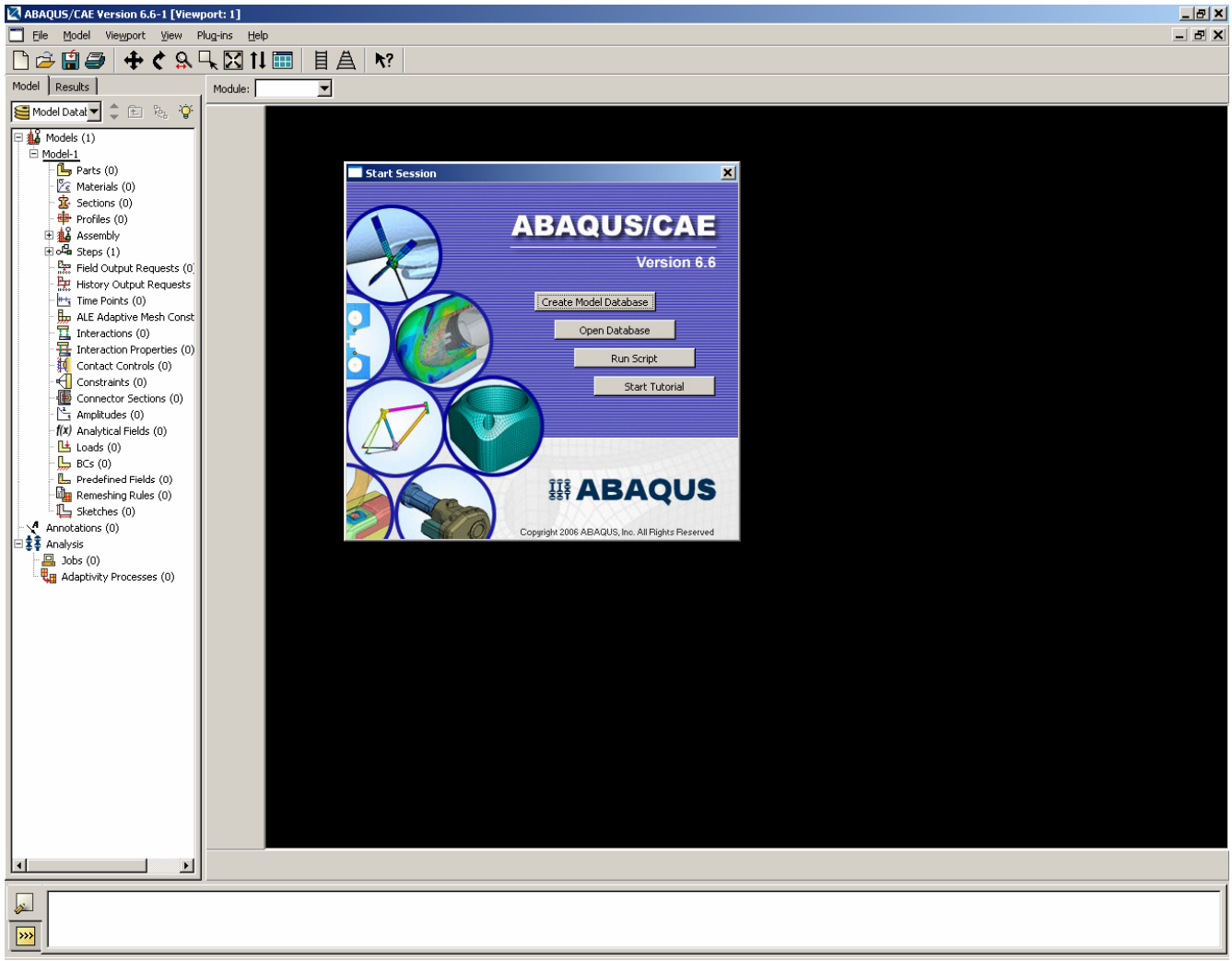


Figure 3.1 ABAQUS main window.

Select ***File*** and than select ***Set Work Directory***. A window that allows you to set the work directory will appear, Figure 3.2. Click ***Select***. A standard window will allow you to select the work directory, Figure 3.3. Press ***OK*** to confirm the selected directory.

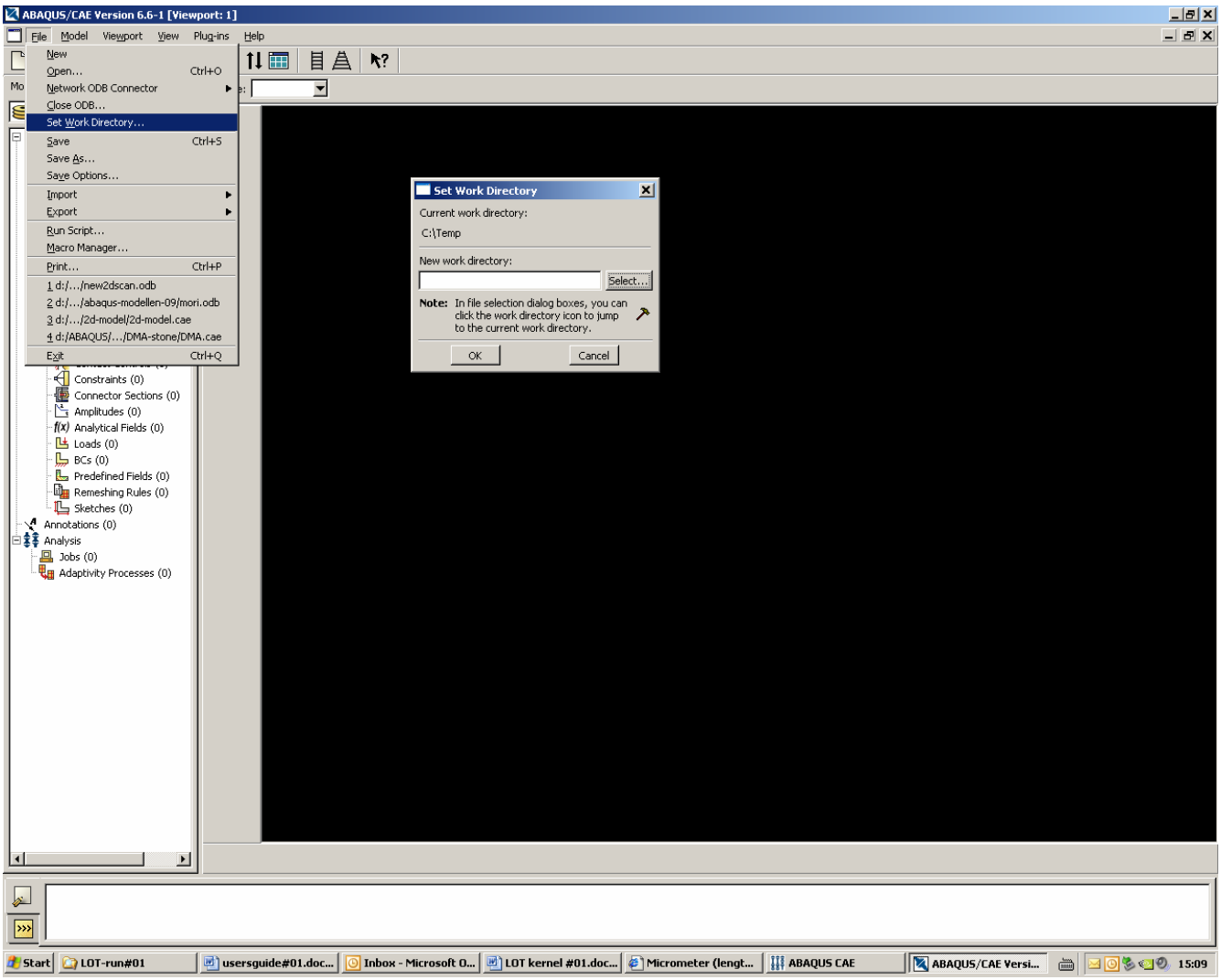


Figure 3.2 Route to set work directory.

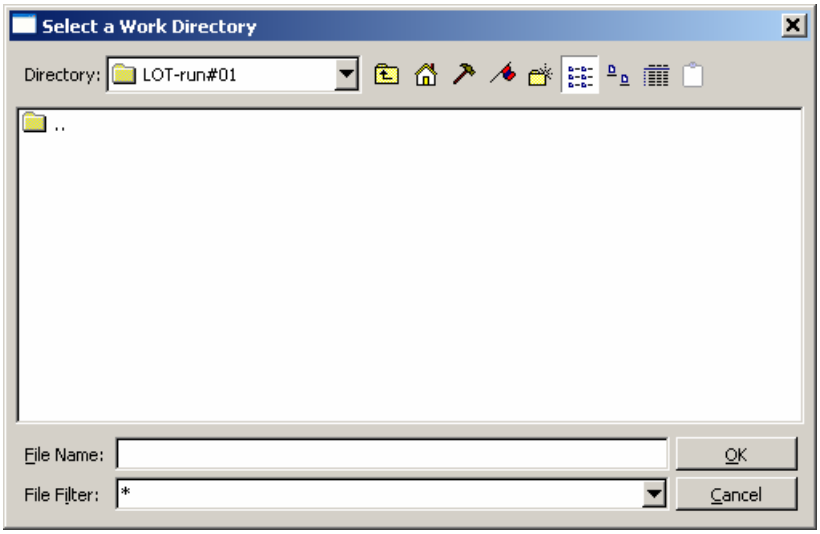


Figure 3.3 Selection of work directory via standard windows.

The work directory is the default directory to which all outputs generated by ABAQUS will be written. With the work directory being set, a LOT run can be started. Hereto double click ***Jobs*** in the lower left corner of the screen. The ***Create Job*** window will now appear.

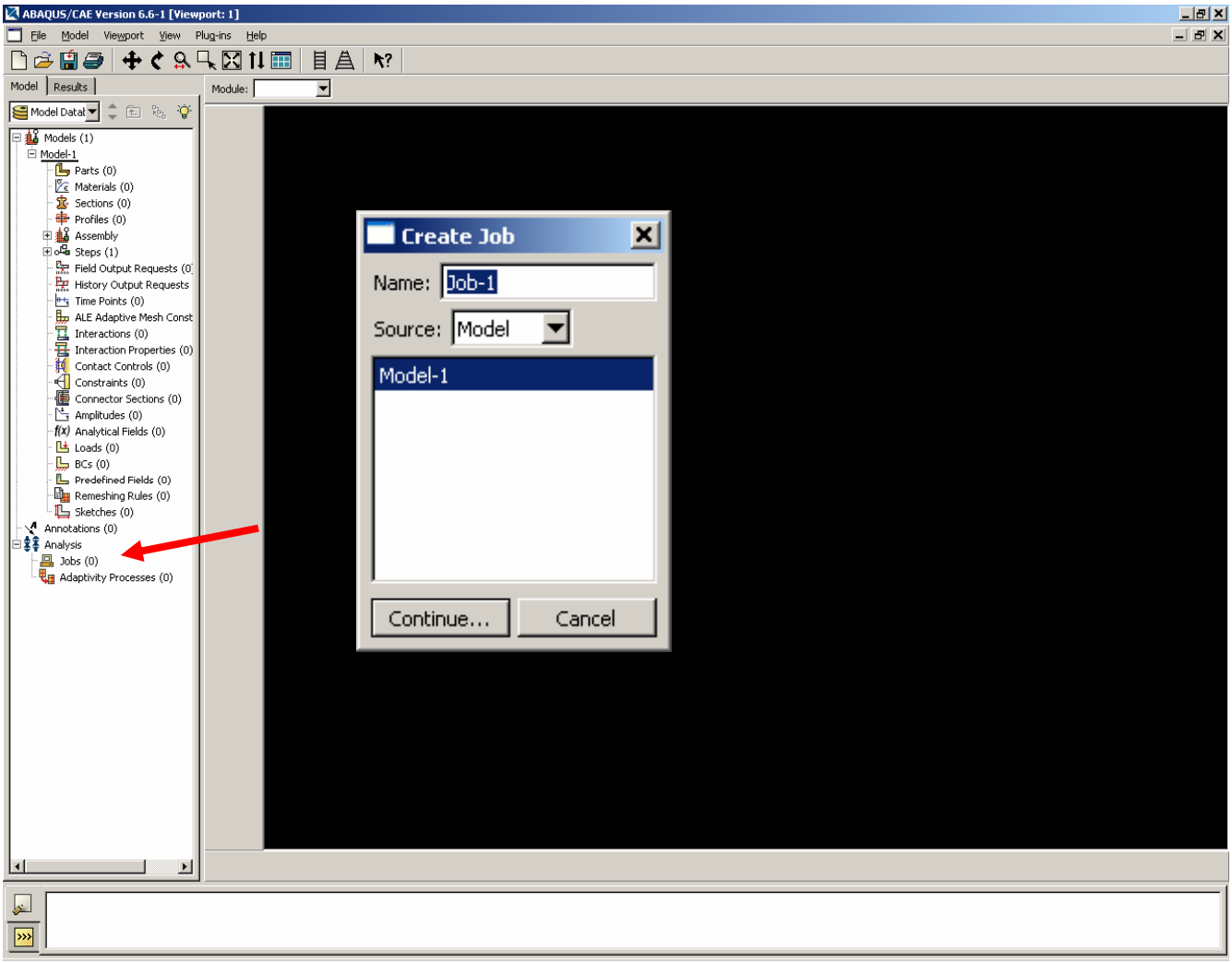


Figure 3.4 The *Create Job* window.

Change the source from ***Model*** into ***Input file*** and then click ***select*** to select the desired input file.

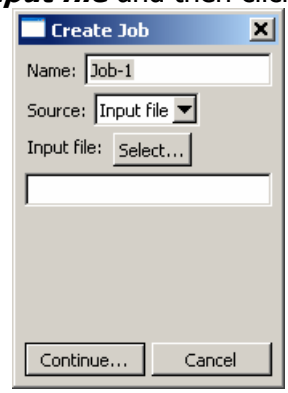


Figure 3.5 The *Create Job* window after selection of *Input file*.

The following window will appear. Click the indicated hammer to jump to the work directory and select the relevant input file. Then Click **OK**. The Job name has now changed into the name of the selected inp-file. When required the job name can be changed into a more appropriate name.
It is stated that the inp-file is the file that you have earlier created with the LOT input generator.

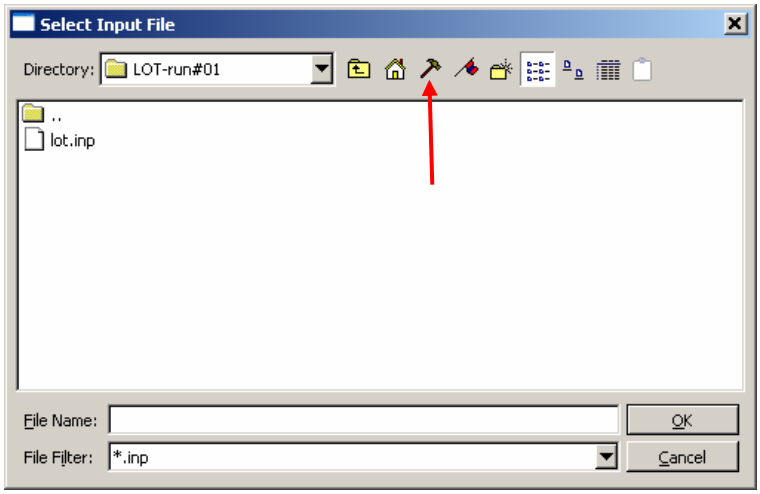


Figure 3.6 The *Select Input File* window.

After the input file is properly selected the ***Edit Job*** window appears. This window has five tabs of which only the ***Memory*** tab may be relevant.

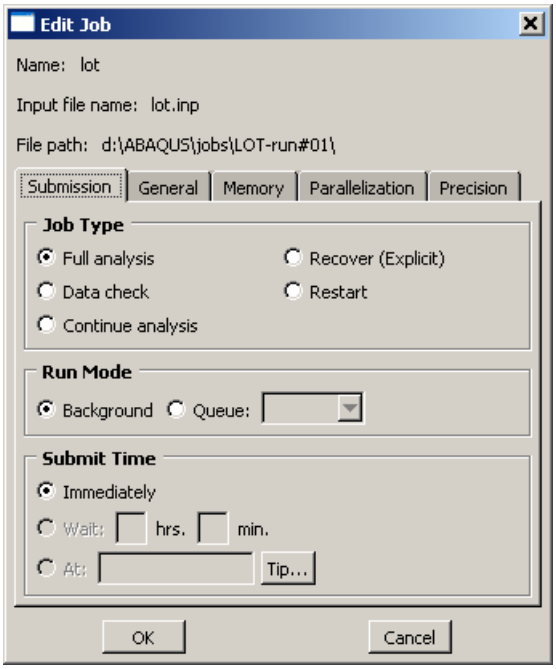


Figure 3.7 The *Edit Job* window.

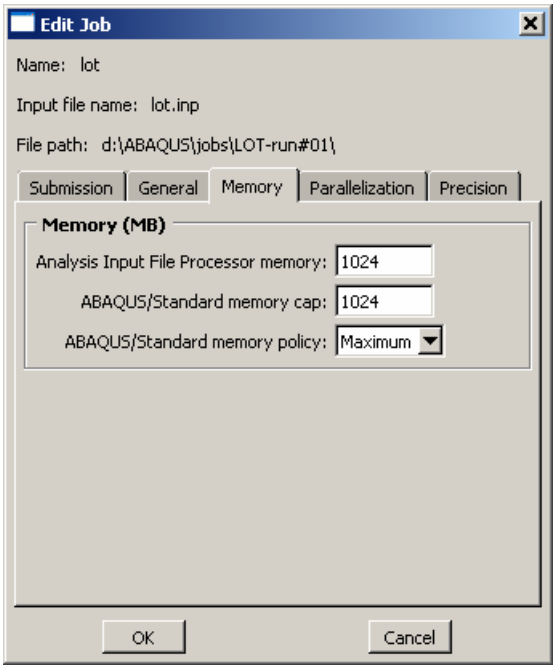


Figure 3.8 The *Edit Job* window after selecting the *Memory* tab.

In the two memory fields increase the maximum amount of memory allocated to ABAQUS. This speeds up the calculation. Secondly put the ABAQUS/Standard memory policy to maximum. By this ABAQUS is given a high memory priority, which can only reduce calculation time.
Close the ***Edit Job*** window by clicking ***OK***.

The Job is now ready to be submitted. To do so, open the job directory in the model-tree by clicking on ***+*** as indicated in figure 3.9. The name of the job that we have created will now appear in the tree. Right click on the job that we want to submit. A drop down menu will appear. Select ***Submit*** in the drop down menu.
The Job is now submitted.

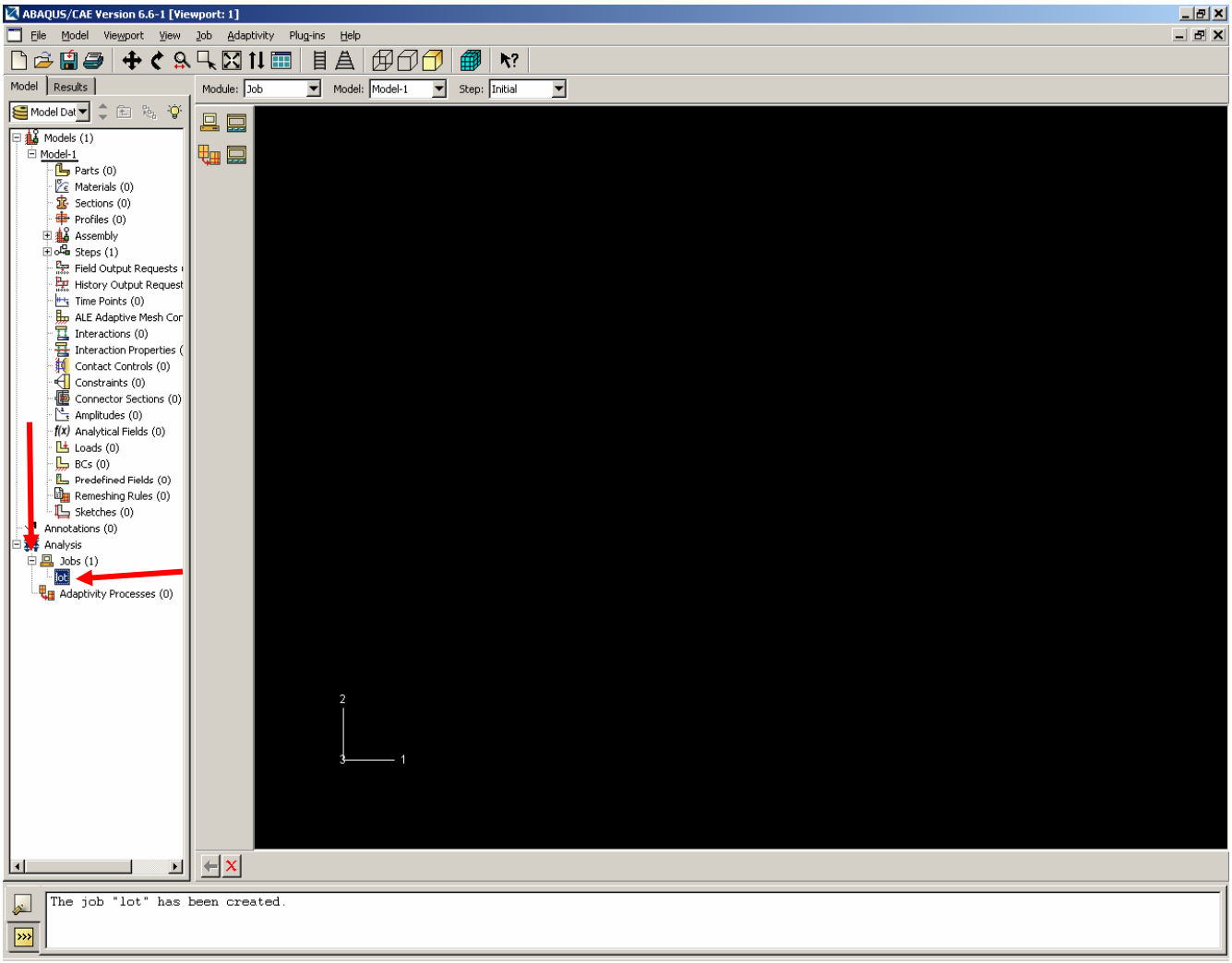


Figure 3.9 Route to submitting a job.

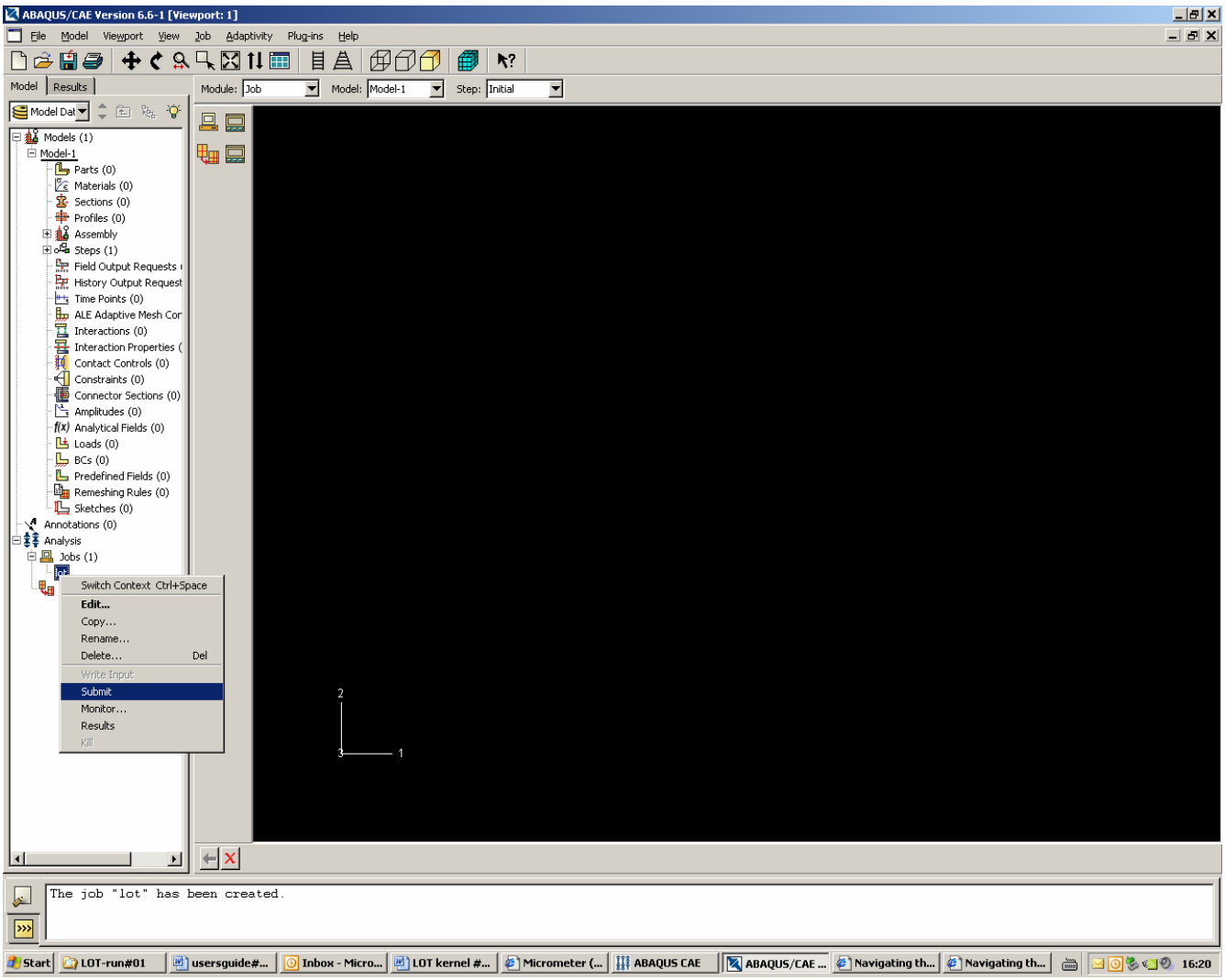


Figure 3.10 To submit select *Submit* in the pull down menu.

ABAQUS will indicate that the selected job is submitted by putting ***(Submitted)*** after the job name in the model tree. This will change into ***(Running)*** when the job is running and finally changes into ***(Completed)*** after completion.

Figure 3.11 Indication of a Running job.

4 POST PROCESSING ABAQUS LOT RUNS

4.1 Introduction

Post processing starts by accessing the ***Visualization*** mode in ABAQUS. Hereto you should select the ***Visualisation*** module, see figure 4.1.

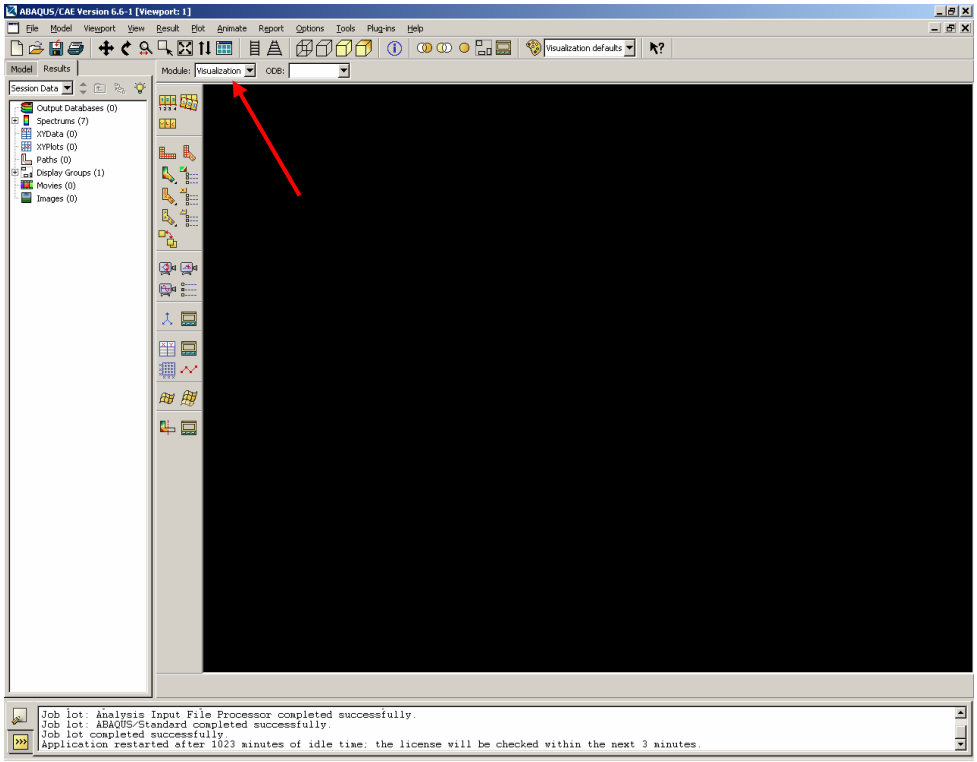

ABAQUS/CAE Version 6.6-1 [Viewport: 1]
File Model Viewport View Result Plot Animate Report Options Tools Plug-ins Help
Visualization defaults
Model Results
Module: Visualization
ODB:
Session Data
Output Databases (0)
Spectrums (7)
XYData (0)
XYPlots (0)
Paths (0)
Display Groups (1)
Movies (0)
Images (0)
Job lot: Analysis Input File Processor completed successfully.
Job lot: ABAQUS/Standard completed successfully.
Job lot completed successfully.
Application restarted after 1023 minutes of idle time; the license will be checked within the next 3 minutes.

Figure 4.1 Select the *Visualization* module as indicated.

After the visualisation module is selected the relevant Output Data Base odb-file should be opened. Hereto the ***Open*** option in the ***File*** pull down menu should be selected, see figure 4.2. After selection of the ***Open*** option select the relevant odb-file.

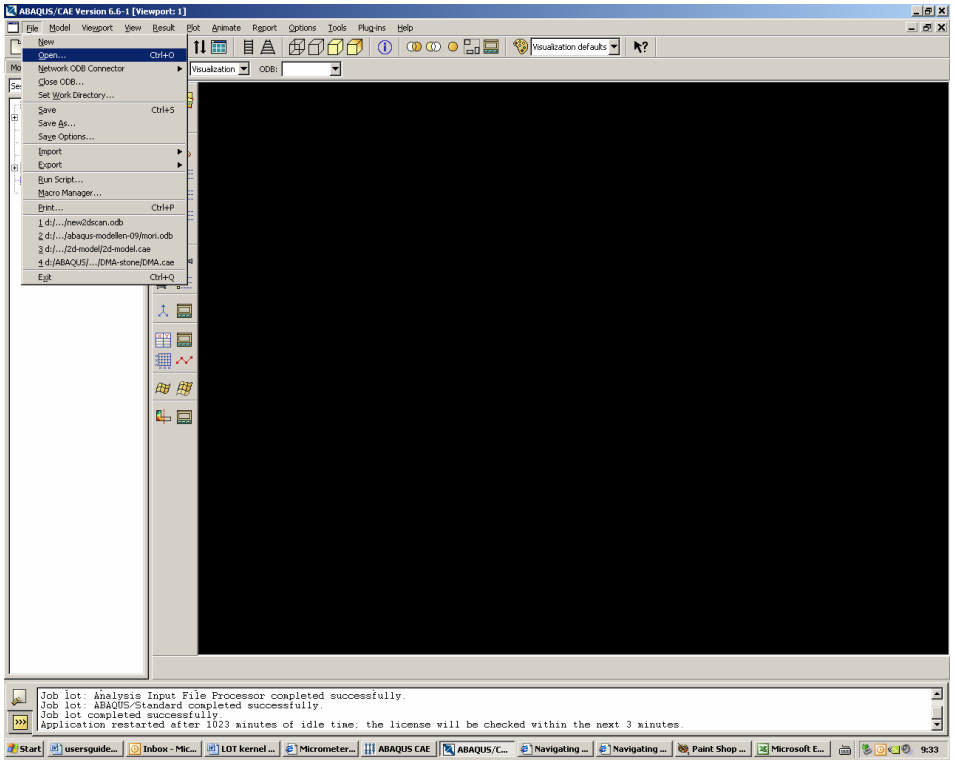

ABAQUS/CAE Version 6.6-1 [Viewport: 1]
File Model Viewport View Result Plot Animate Report Options Tools Plug-ins Help
New
Open... Ctrl+O
Network ODB Connector
Close ODB...
Set Work Directory...
Save Ctrl+S
Save As...
Save Options...
Import
Export
Run Script...
Macro Manager...
Print... Ctrl+P
1 d:/.../new2dscan.odb
2 d:/.../abaqus-modellen-09/mori.odb
3 d:/.../2d-model/2d-model.cae
4 d:/ABAQUS/.../DMA-stone/DMA.cae
Exit Ctrl+Q
Visualization defaults
Visualization
ODB:
Job lot: Analysis Input File Processor completed successfully.
Job lot: ABAQUS/Standard completed successfully.
Job lot completed successfully.
Application restarted after 1023 minutes of idle time: the license will be checked within the next 3 minutes.
Start usersguide... Inbox - Mic... LOT kernel ... Micrometer... ABAQUS CAE ABAQUS/C... Navigating ... Navigating ... Paint Shop ... Microsoft E... 9:33

Figure 4.2 Route to open the relevant odb-file.

The screen will now show the model to which the selected odb-file refers. For further processing it is advised to change the screen background colour to a lighter colour. Hereto select ***Graphic Options*** in the ***View*** pull down menu, Figure 4.3.

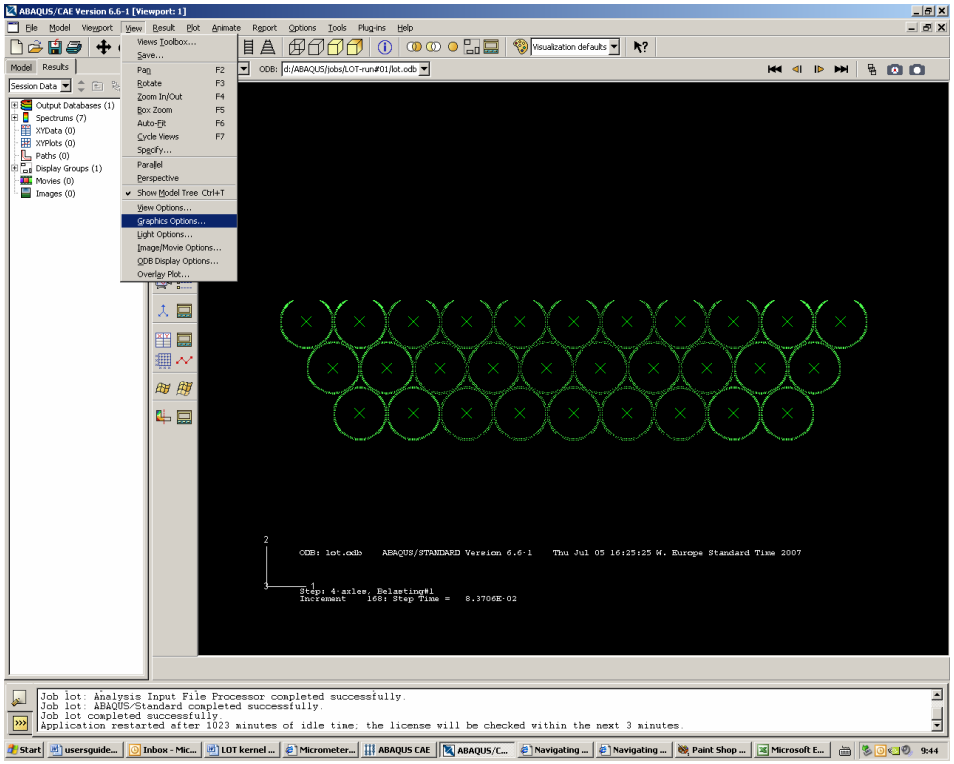

ABAQUS/CAE Version 6.6-1 [Viewport: 1]
File Model Viewport View Result Plot Animate Report Options Tools Plug-ins Help
Views Toolbox...
Save...
Pan F2
Rotate F3
Zoom In/Out F4
Box Zoom F5
Auto-Fit F6
Cycle Views F7
Specify...
Parallel
Perspective
Show Model Tree Ctrl+T
View Options...
Graphics Options...
Light Options...
Image/Movie Options...
ODB Display Options...
Overlay Plot...
Model Results
Session Data
Output Databases (1)
Spectrums (7)
XYData (0)
XYPlots (0)
Paths (0)
Display Groups (1)
Movies (0)
Images (0)
ODB: d:/ABAQUS/jobs/LOT-run#01/lot.odb
ODB: lot.odb ABAQUS/STANDARD Version 6.6-1 Thu Jul 05 16:25:25 W. Europe Standard Time 2007
Step: 4-axles, Belasting#1
Increment 168: Step Time = 8.3706E-02
Job lot: Analysis Input File Processor completed successfully.
Job lot: ABAQUS/Standard completed successfully.
Job lot completed successfully.
Application restarted after 1023 minutes of idle time: the license will be checked within the next 3 minutes.
Start usersguide... Inbox - Mic... LOT kernel ... Micrometer... ABAQUS CAE ABAQUS/C... Navigating ... Navigating ... Paint Shop ... Microsoft E... 9:44

Figure 4.3 Route to the *Graphic Options* window.

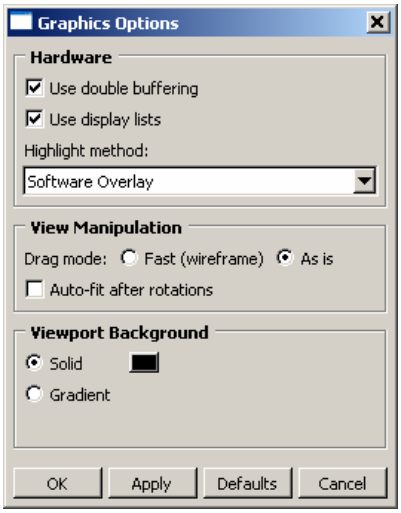


Figure 4.4 The *Graphic Options* window.

In the ***Graphic Options*** window click the black button and select a lighter background colour. Close the window by clicking ***OK***.

To estimate the life expectancy of the adhesive zones in the modelled PA, insight into the stress-time signal in these zones is required. The damage that is introduced by this stress-time signal can be obtained by application of the adhesive zone fatigue damage model discussed in [1] and [2].
To estimate the life expectancy of the mortar bridges in the modelled PA insight into the stress-strain signal in the mortar is required. The damage that is introduced by this stress-strain signal can be obtained by application of the mortar dissipated energy fatigue damage model discussed in [1] and [2].
In the following sections it is discussed how the required signals are extracted from ABAQUS and written to ASCII-file. The obtained ASCII-files can of course be imported into Excel – or any other general purpose program – that will then allow application of the fatigue models discussed in [1] and [3]. Application of these models on ABAQUS output is beyond the scope of this report and not discussed here.

4.2 2D idealised model

4.2.1 Adhesive zone damage

Most accurate results are obtained in the central part of the model. For this reason the contact areas of the central surface stone should be considered for post processing. To obtain the stress-time signals in the adhesive zones around the central stone first of all click ***XY-data*** and select ***ODB field output*** in the window that emerges, Figure 4.5.

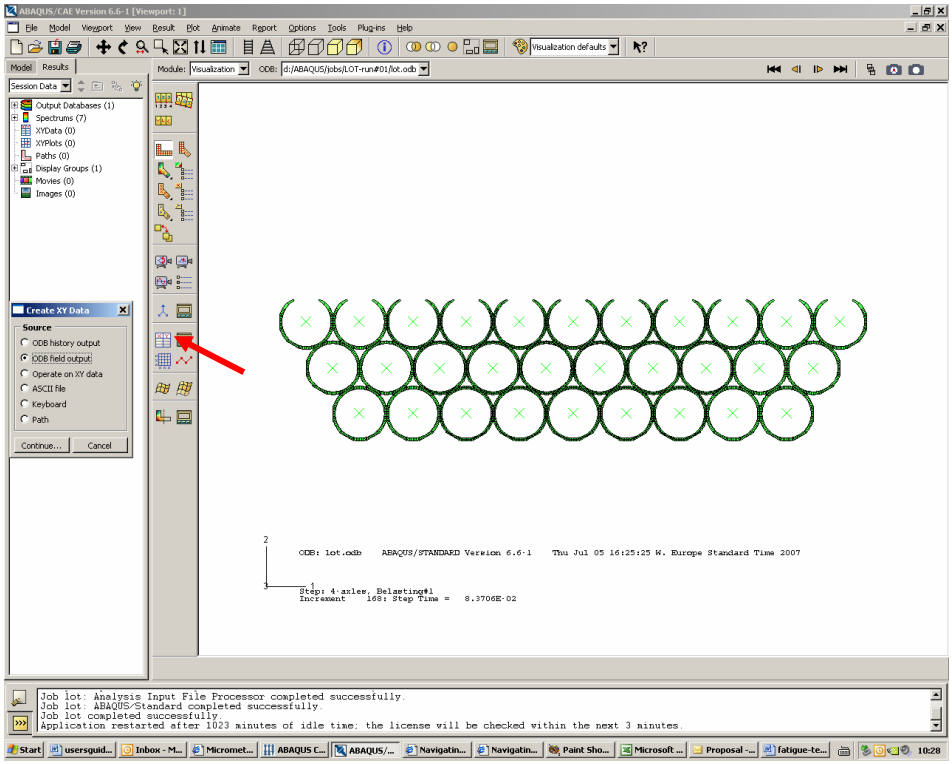

Create XY Data
Source
ODB history output
ODB field output
Operate on XY data
ASCII file
Keyboard
Path
Continue...
Cancel
ODB: lot.odb ABAQUS/STANDARD Version 6.6-1 Thu Jul 05 16:25:25 W. Europe Standard Time 2007
Step: 4-axles, Belasting#1
Increment 168: Step Time = 8.3706E-02
Job lot: Analysis Input File Processor completed successfully.
Job lot: ABAQUS/Standard completed successfully.
Job lot completed successfully.
Application restarted after 1023 minutes of idle time; the license will be checked within the next 3 minutes.

Figure 4.5 Click *XY-data* and select *OBD field output*.

After clicking ***Continue*** the ***XY Data from ODB Field Output*** window is opened. In this window select ***Unique Nodal*** as position. Select ***S22*** and ***S12*** as the Stress Components of interest and open the ***Elements/Nodes*** tab.

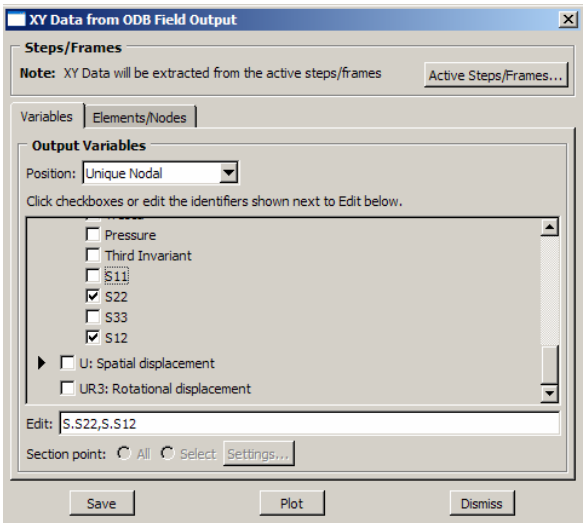


Figure 4.6 The *XY Data from ODB Field Output* window before opening the *Element/Nodes* tab.

Select ***Node Labels*** as the ***Method*** and select ***Part-6*** under ***Part instance***. Enter the desired node number in the ***Node labels*** field and select ***Highlight items in viewport***, see Figure 4.7. When desired several node numbers can be entered in the ***Node labels*** field. Hereto node labels need to be separated by comma-blank.

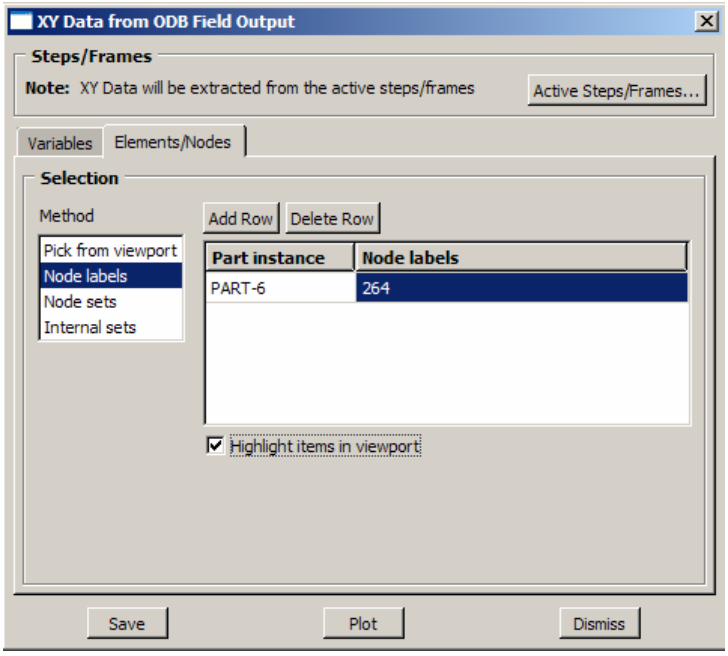


Figure 4.7 Window for selection of nodes for which output is generated.

In the 2D idealised model each of the surface stones has 4 stone-stone contacts. To investigate the state of stress in the adhesive zone, the node sitting in the centre of each contact region needs to be selected. For PART-6, i.e. the central surface stone the following nodes are of interest.

Table 4.1 Relevant nodes for adhesive zone analyses with the 2D idealised model.

Interface	Node
CR-1	Part-6: 264
CR-2	Part-6: 454
CR-3	Part-6: 644
CR-4	Part-6: 20

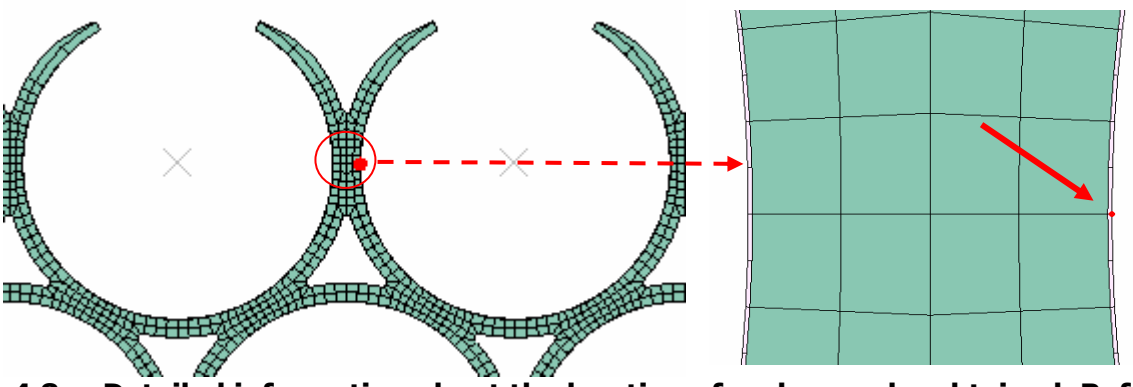


Figure 4.8 Detailed information about the location of nodes can be obtained. Reference is made the ABAQUS users manual [5].

After all desired information is entered in the ***XY Data from ODB Field Output*** click ***Plot*** and close the window. Now select the ***XY …*** option in the ***Report*** pull down menu.

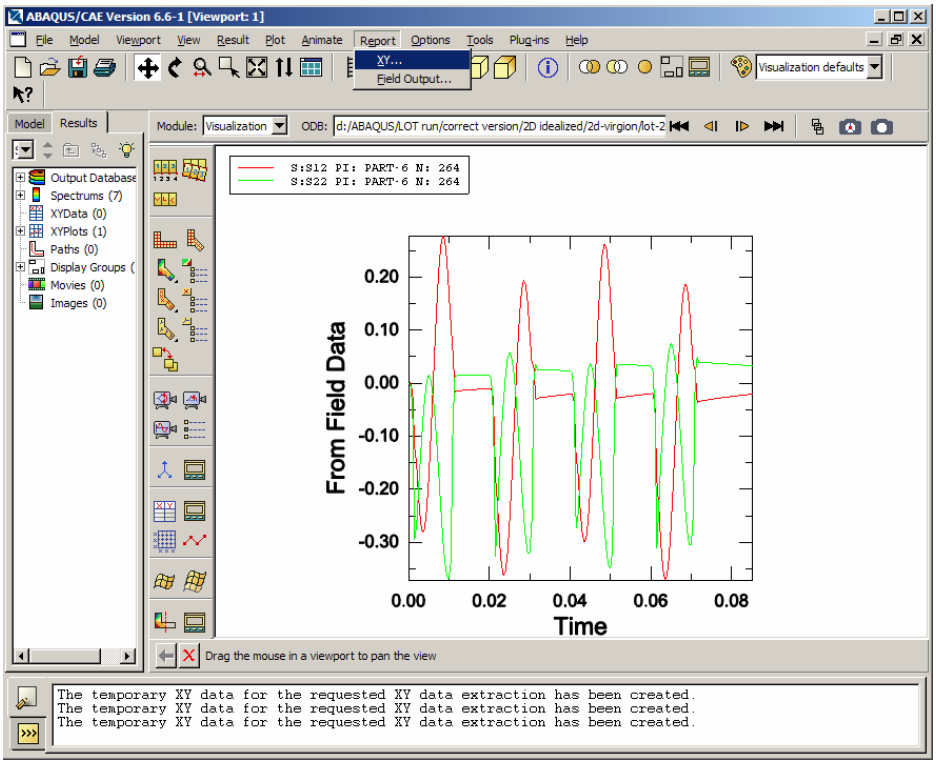


Figure 4.9 Select *XY …* to export data to ASCII-file.

In the widow that appears first select ***XY plot in current viewport***, then select stress components ***S12*** and ***S22***.

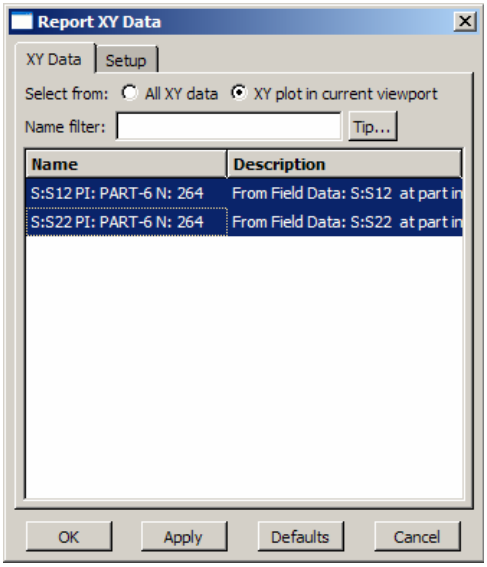


Figure 4.10 Route to ASCII-file export.

Now click the ***Setup*** tab and enter the name of the ASCII-file to which the data is to be exported. Finally choose options as per Figure 4.11 and click ***Apply***.

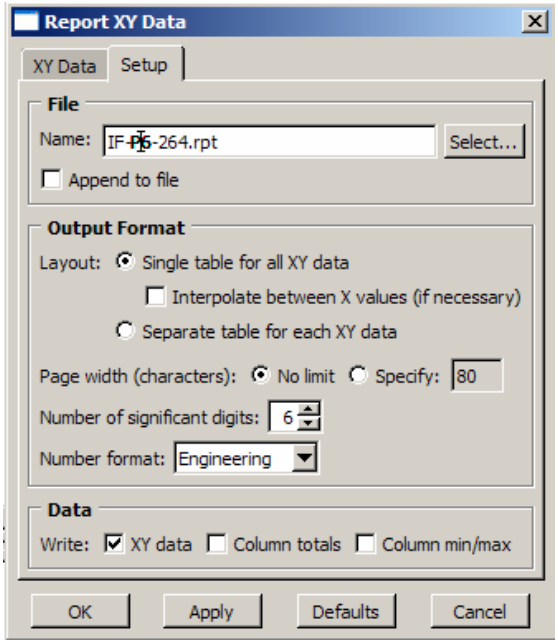


Figure 4.11 Route to ASCII-file export.

An ABAQUS report file (*.rpt) is now generated in the work directory. The report file contains the desired computational data, which can be imported into Excel for further data analysis. The stress components for interface zone are as follows:

S22 ***= Direct through-thickness stress***
S12 ***= Transverse shear stress***

4.2.2 Mortar damage

The process of exporting the relevant stress-strain signals to which the mortar bridges are subjected is similar to the process of exporting adhesive zone stress-signals. For this reason reference is made to section 4.2.1 for detailed information. In this section the process is briefly discussed.

The relevant nodes in the 2D idealised models are listed in table 4.2.

Table 4.2 Relevant nodes for motar fatigue analyses

Interface	Node
CR-1	Part 6: 414
CR-2	Part 16: 365
CR-3	Part 17: 555
CR-4	Part 7: 414

First open the relevant output data base file (*.odb) in ABAQUS visualization mode. Then proceed to the ***XY Data from ODB field output*** window as described in section 4.2.1. Again select ***Unique Nodal***. Now select the Stress components ***S11***, ***S22*** and ***S12***, Figure 4.12. Also select Logarithmic strain components ***LE11***, ***LE22*** and ***LE12***, Figure 4.13.
Depending on the capacity of the computer exporting stress and strain signals in one go may be troublesome. If required the data can be exported in two separate actions.

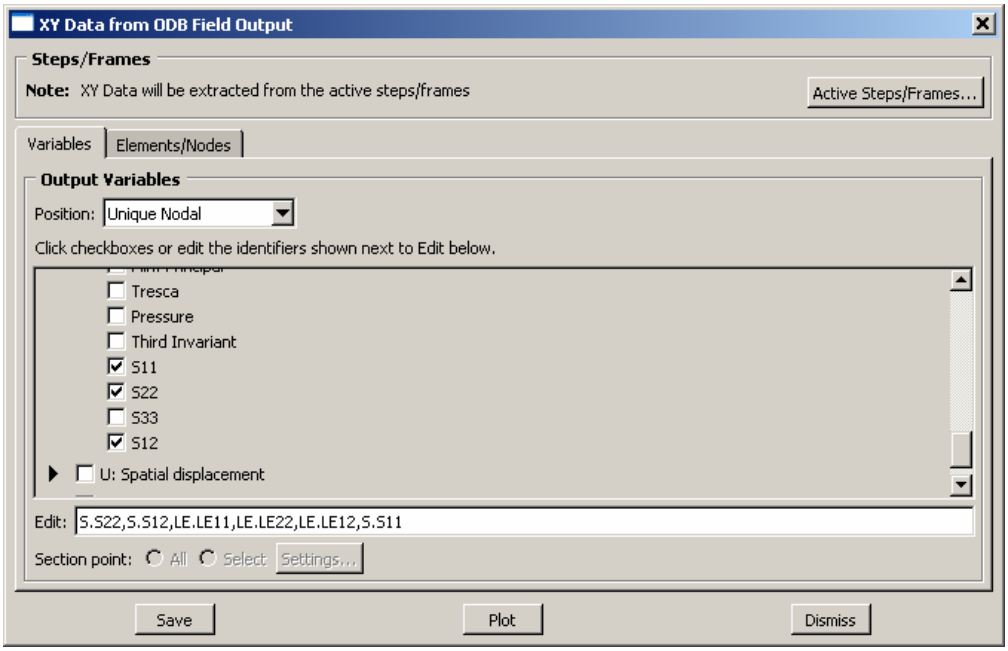


Figure 4.12 Route to ASCII-file export, selection of Stress components.

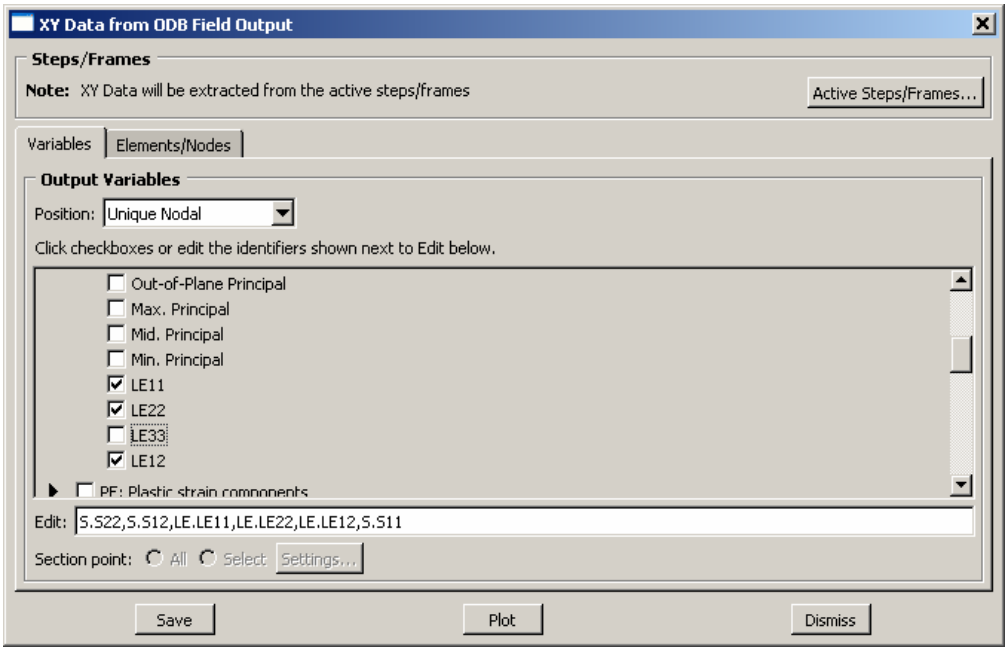


Figure 4.13 Route to ASCII-file export, selection of Strain components.

Now click the ***Element/Nodes*** tab and input the desired node, see Table 4.2 and Figure 4.14. Check the position of the node as per figure 4.15.

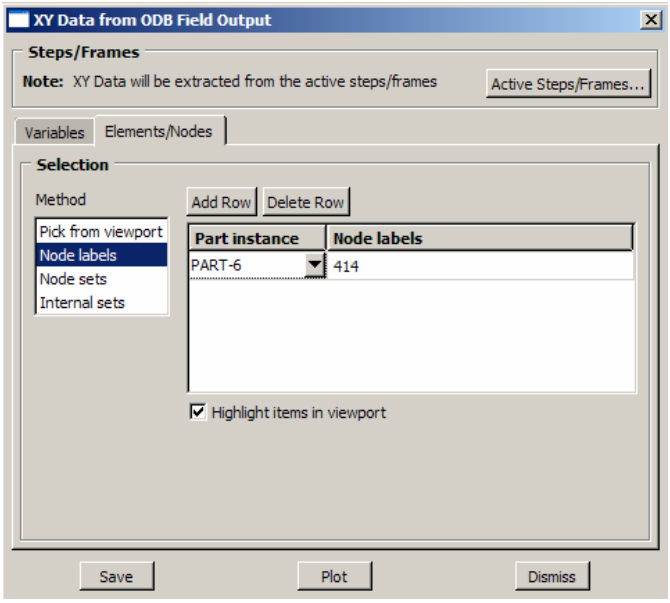


Figure 4.14 Route to ASCII-file export, input of desired node.

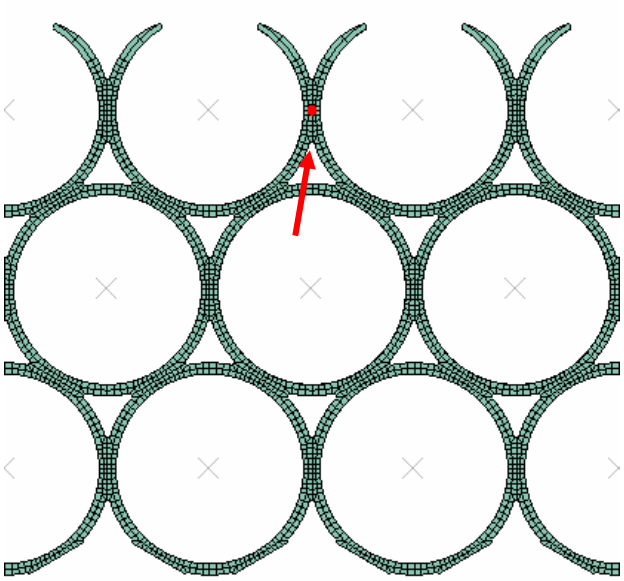


Figure 4.15 Route to ASCII-file export, indication of the location of the node.

Finally click ***Plot***. The screen now changes and becomes as shown in Figure 4.16. Proceed to the ***Report XY data*** window, Figure 4.17, and export the data to ASCII-file.

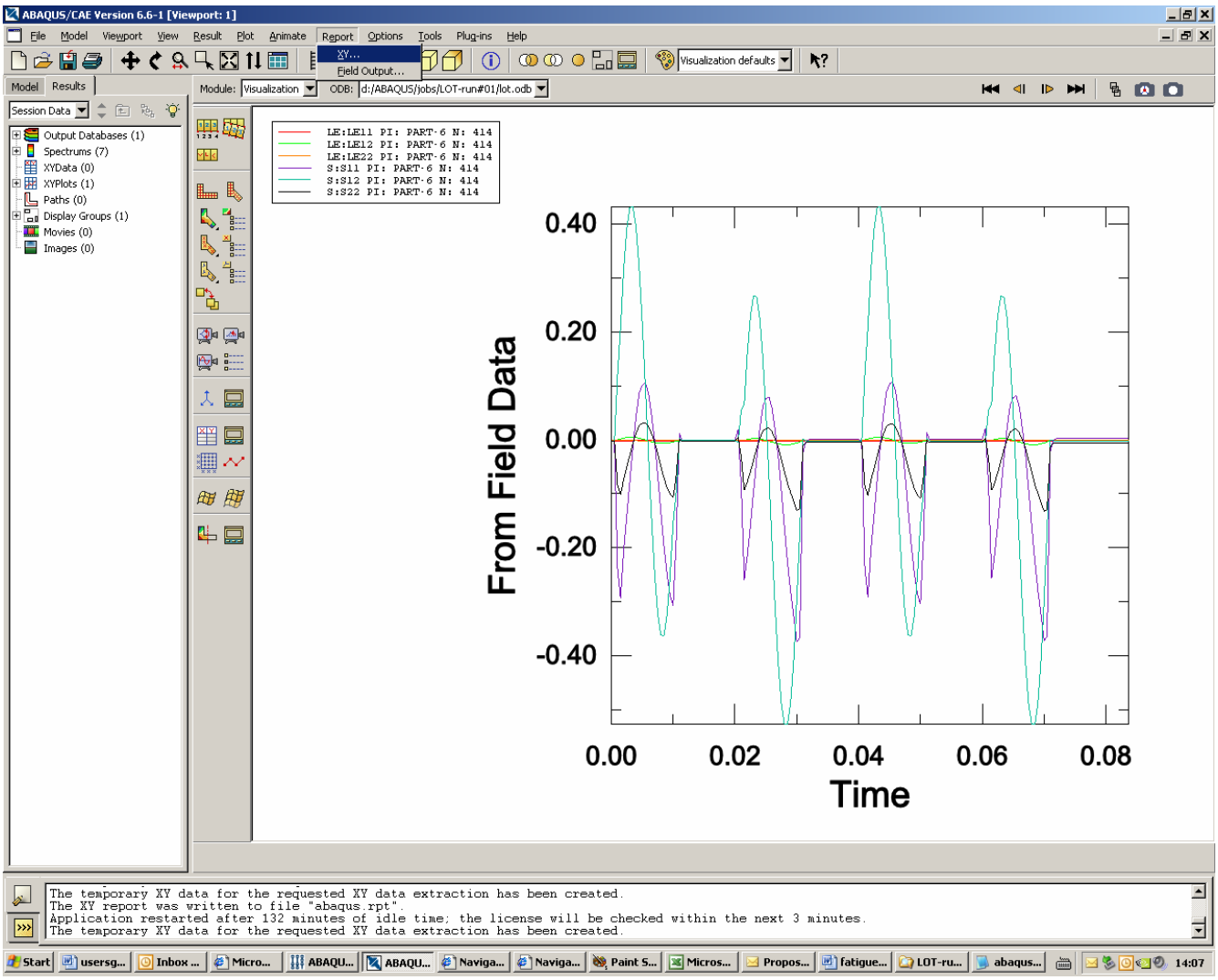


Figure 4.16 Route to ASCII-file export, opening *Report XY data* window.

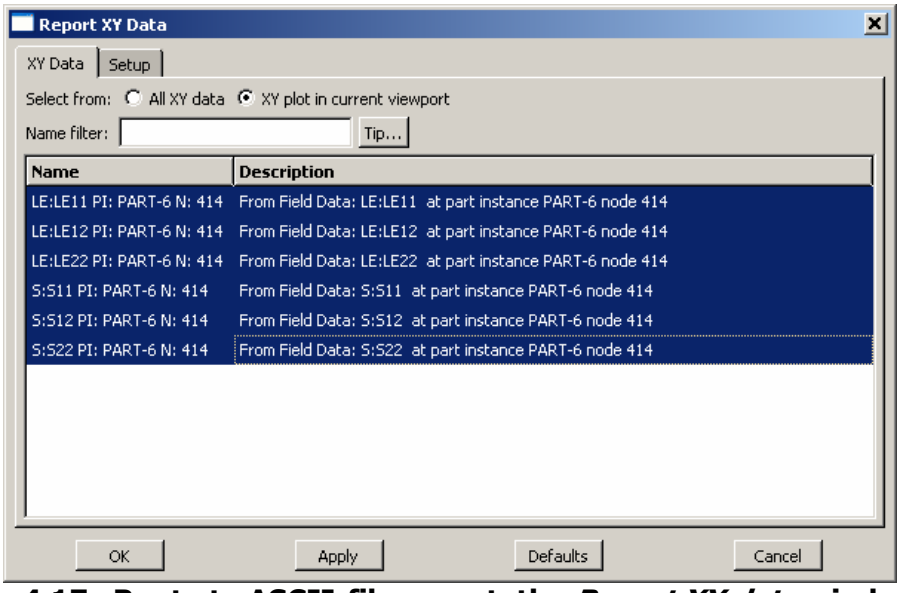

Report XY Data
XY Data
Setup
Select from:
All XY data
XY plot in current viewport
Name filter:
Tip...
Name
Description
LE:LE11 PI: PART-6 N: 414
From Field Data: LE:LE11 at part instance PART-6 node 414
LE:LE12 PI: PART-6 N: 414
From Field Data: LE:LE12 at part instance PART-6 node 414
LE:LE22 PI: PART-6 N: 414
From Field Data: LE:LE22 at part instance PART-6 node 414
S:S11 PI: PART-6 N: 414
From Field Data: S:S11 at part instance PART-6 node 414
S:S12 PI: PART-6 N: 414
From Field Data: S:S12 at part instance PART-6 node 414
S:S22 PI: PART-6 N: 414
From Field Data: S:S22 at part instance PART-6 node 414
OK
Apply
Defaults
Cancel

Figure 4.17 Route to ASCII-file export, the *Report XY data* window.

For determination of the dissipated energy the following signals are considered.

LE11 vs S11: ***Strain 11 and stress 11 components***
LE12 vs S12: ***Strain 12 and stress 12 components***
LE22 vs S22: ***Strain 22 and stress 22 components***

4.3 3D idealized model

4.3.1 Adhesive zone damage

First open the relevant odb-file in the ABAQUS visualization module as per section 4.1. In this module relevant adhesive stress signals can be exported similarly as described in detail in section 4.2.1. The procedure is discussed briefly hereafter.

The modelled stones in the top layer each have 9 stone-stone contacts. To investigate the state of stress in the adhesive zones surrounding the central stone the nodes indicated in table 4.3 are relevant. Each of these nodes sits in the centre of the 9 contact areas.

Table 4.3 Information about selected nodes/elements for IF analysis

Interface	Node	Element
CR 1	Top-1: 4	157
CR 2	Top-1: 156	182
CR 3	Top-1: 288	207
CR 4	Top-1: 420	232
CR 5	Top-1: 552	257
CR 6	Top-1: 684	333
CR 7	Top-1: 969	1009
CR 8	Top-1: 1333	1051
CR 9	Top-1: 1673	1093

To export adhesive zone stress data first select ***XY-data*** as indicated in Figure 4.18. Select ***odb field output*** and indicate ***unique nodal*** in the ***XY Data from ODB field output*** window. Then select the stress components ***S33***, ***S13*** and ***S23*** as per figure 4.19.

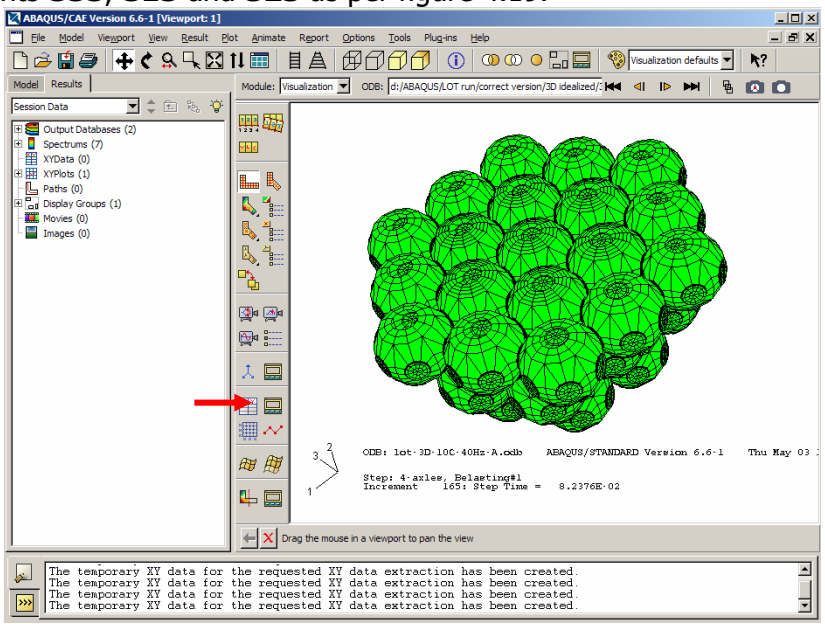

ABAQUS/CAE Version 6.6-1 [Viewport: 1]
File Model Viewport View Result Plot Animate Report Options Tools Plug-ins Help
Model Results
Session Data
Output Databases (2)
Spectrums (7)
XYData (0)
XYPlots (1)
Paths (0)
Display Groups (1)
Movies (0)
Images (0)
Module: Visualization
ODB: d:/ABAQUS/LOT run/correct version/3D idealized/
Drag the mouse in a viewport to pan the view
The temporary XY data for the requested XY data extraction has been created.
The temporary XY data for the requested XY data extraction has been created.
The temporary XY data for the requested XY data extraction has been created.
The temporary XY data for the requested XY data extraction has been created.

Figure 4.18 Route to ASCII-file export of adhesive zone stress.

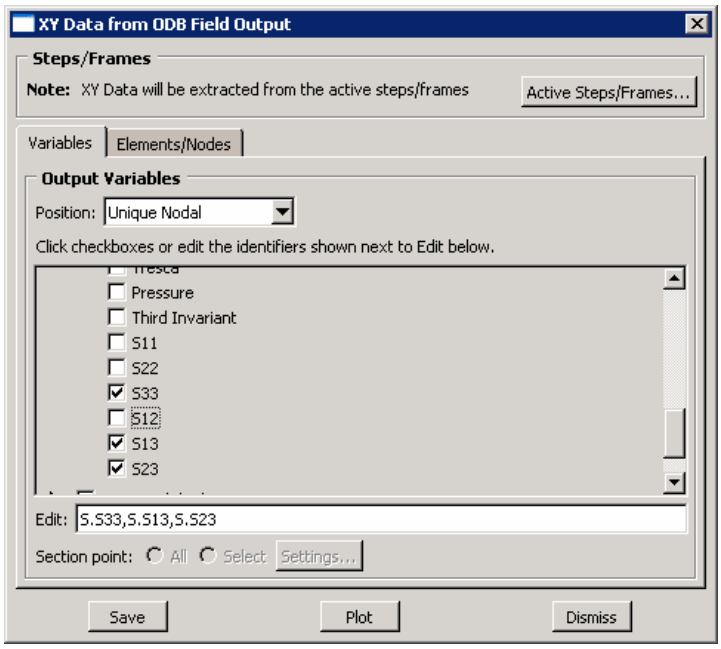


Figure 4.19 Select the stress component *S33*, *S13* and *S23* for adhesive zone analyses.

After the selection of the stress components click the ***Elements/Nodes*** tab and put in the desired node as per Table 4.3, then click ***Plot***. Hereafter proceed to the ***report*** drop down menu for export of data to ASCII-file, see figure 4.20. Select all plotted data for export, Figure 4.21.

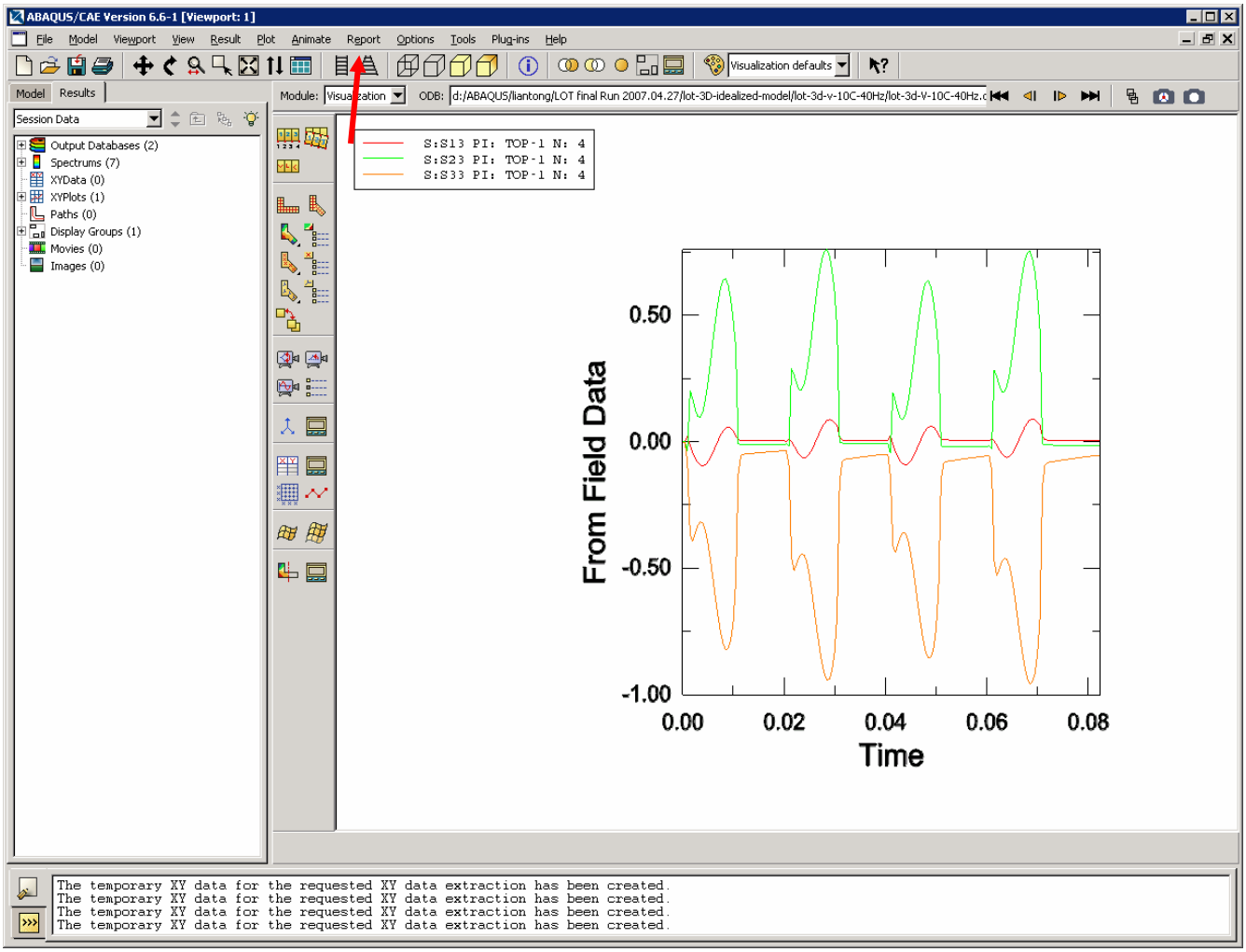


Figure 4.20 Route to ASCII-file export, opening *Report XY data* window.

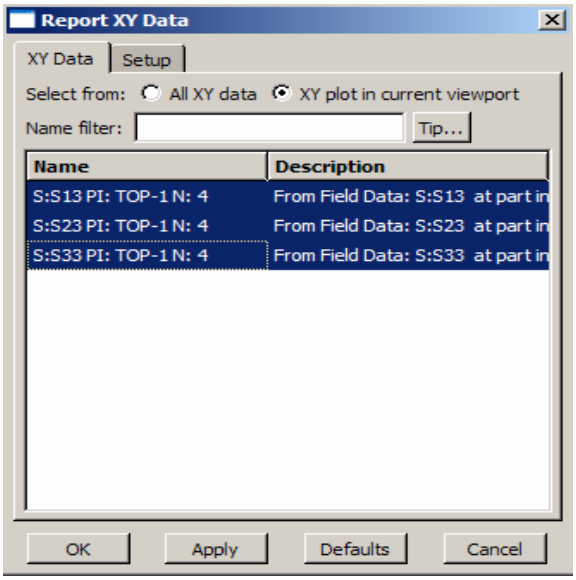


Figure 4.21 Route to ASCII-file export, the *Report XY data* window.

Please give the export file a name by entering the ***Setup*** tab. Thereafter click ***OK***. An ABAQUS report file (*.rpt) is now generated in the work directory. The report file contains the desired computational data, which can be imported into Excel for further data analysis. The stress components for the adhesive zone are as follows:

S33 = Direct through-thickness stress
S13, S23 = Shear stresses in the local in-plane directions

4.3.2 Mortar damage

The process of exporting the relevant stress-strain signals for the mortar bridges is similar to the process of exporting other data signals. For this reason reference is made to section 4.2.1 for detailed information. In this section the process of exporting mortar data is briefly discussed.

The relevant nodes in the 3D idealised models are listed in table 4.4.

Table 4.4 Relevant nodes for motar fatigue analyses

Mortar in contact region	Node. No.
CR 1	Top-1: 1
CR 2	Top-1: 153
CR 3	Top-1: 285
CR 4	Top-1: 417
CR 5	Top-1: 549
CR 6	Top-1: 681
CR 7	Top-1: 966
CR 8	Top-1: 1330
CR 9	Top-1: 1670

First open the relevant output data base file (*.odb) in ABAQUS visualization mode. Then proceed to the ***XY Data from ODB field output*** window as described in section 4.2.1. Again select ***Unique Nodal***. Now select the Stress components ***S11***, ***S22***, ***S33***, ***S12***, ***S13*** and ***S23***, Figure 4.22.

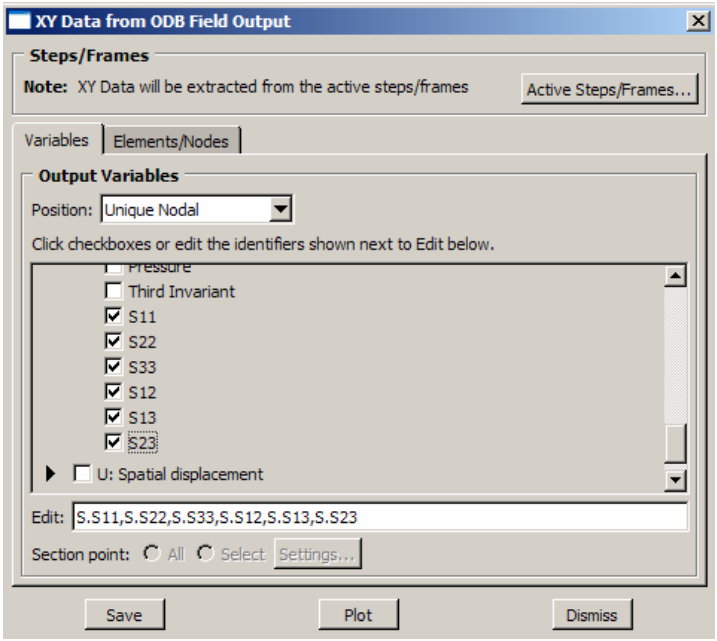


Figure 4.22 Route to ASCII-file export, selection of Stress components.

Now click the ***Element/Nodes*** tab and input the desired node, see Table 4.4 and Figure 4.23. Check the position of the node if desired.

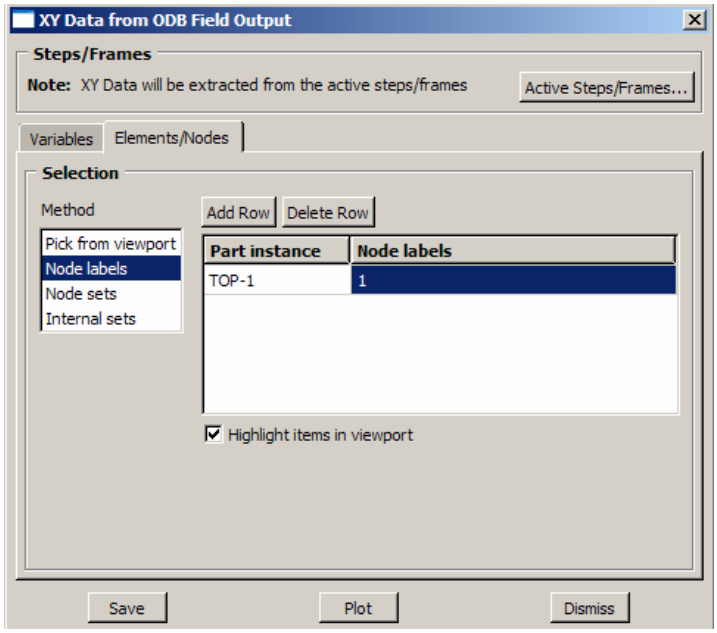


Figure 4.23 Route to ASCII-file export, input of desired node.

Finally click ***Plot***. The screen now changes and becomes as shown in Figure 4.24. Proceed to the ***Report XY data*** window, Figure 4.25, and export the data to ASCII-file.

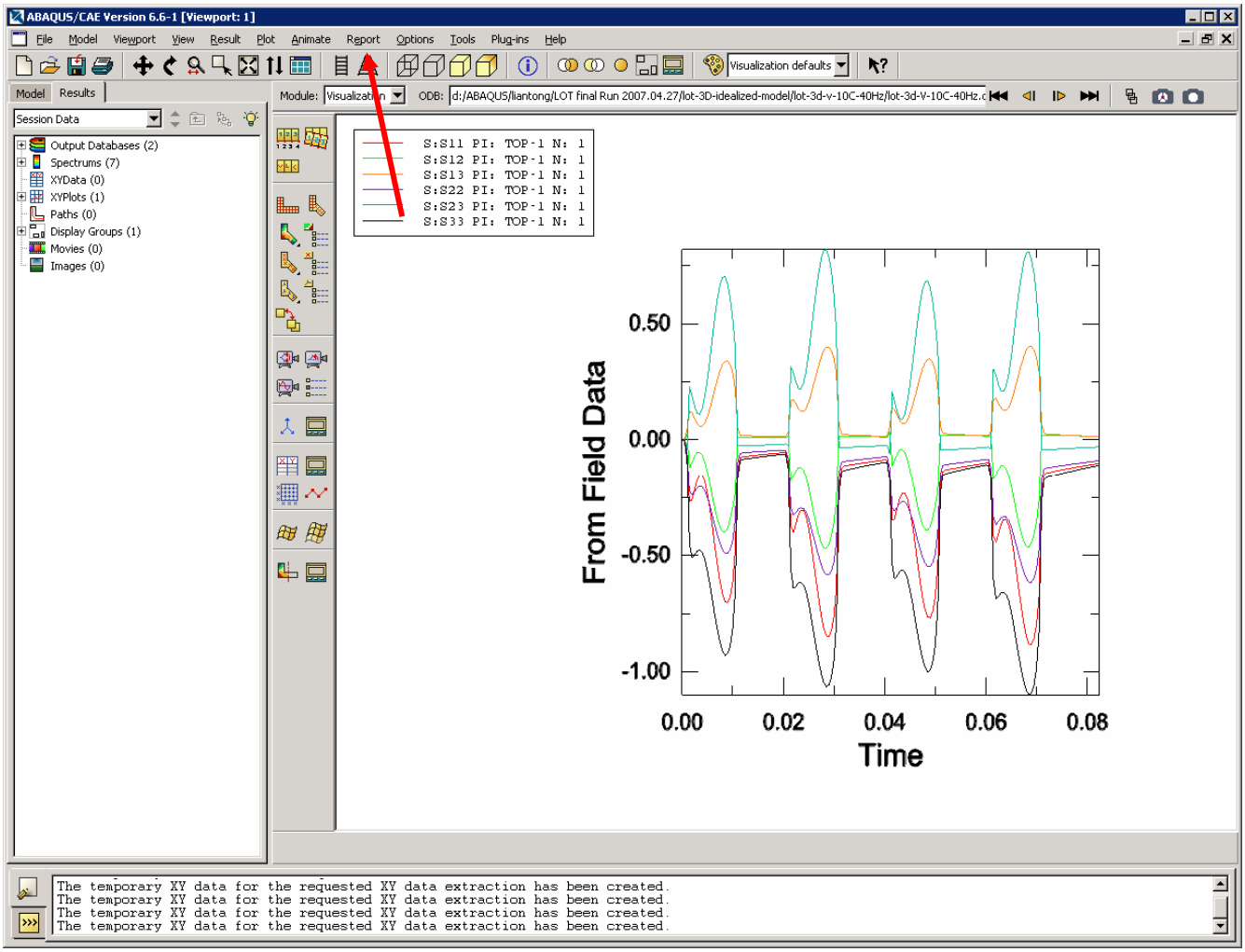

ABAQUS/CAE Version 6.6-1 [Viewport: 1]
File Model Viewport View Result Plot Animate Report Options Tools Plug-ins Help
Visualization defaults
Model Results
Session Data
Output Databases (2)
Spectrums (7)
XYData (0)
XYPlots (1)
Paths (0)
Display Groups (1)
Movies (0)
Images (0)
S:S11 PI: TOP-1 N: 1
S:S12 PI: TOP-1 N: 1
S:S13 PI: TOP-1 N: 1
S:S22 PI: TOP-1 N: 1
S:S23 PI: TOP-1 N: 1
S:S33 PI: TOP-1 N: 1
From Field Data
0.50
0.00
-0.50
-1.00
0.00
0.02
0.04
0.06
0.08
Time
The temporary XY data for the requested XY data extraction has been created.
The temporary XY data for the requested XY data extraction has been created.
The temporary XY data for the requested XY data extraction has been created.
The temporary XY data for the requested XY data extraction has been created.

Figure 4.24 Route to ASCII-file export, opening *Report XY data* window.

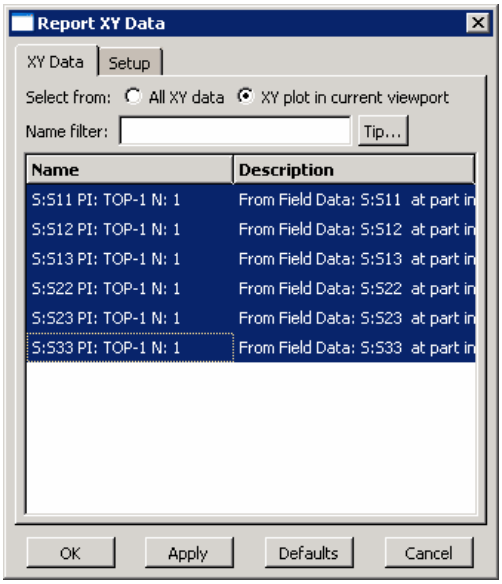


Figure 4.25 Route to ASCII-file export, the *Report XY data* window.

Hereafter repeat the process for the LE strain components ***LE11***, ***LE22***, ***LE33***, ***LE12***, ***LE13*** and ***LE23***, see Figure 4.26.

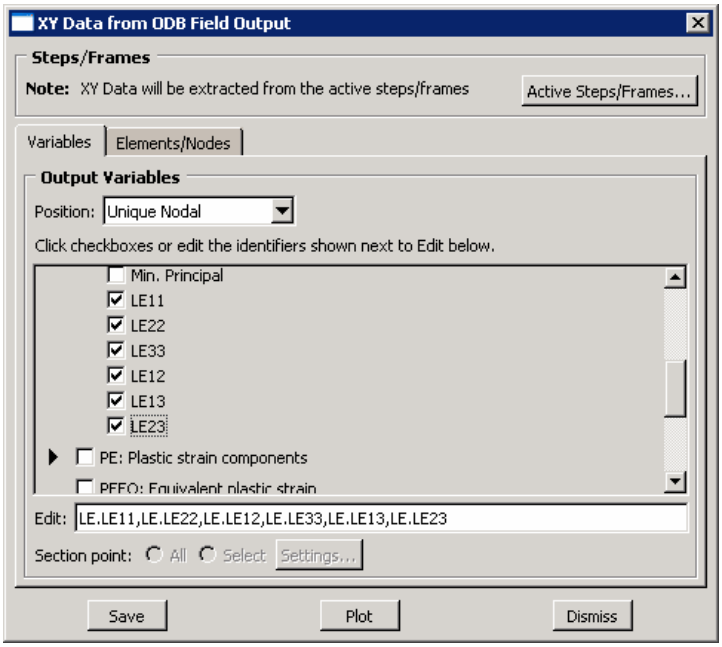


Figure 4.26 Route to ASCII-file export, selection of Stress components.

Now click the ***Element/Nodes*** tab and input the desired node, see Table 4.4. Check the position of the node if desired. Finally click ***Plot***. The screen now changes and becomes as shown in Figure 4.27. Proceed to the ***Report XY data*** window, Figure 4.28, and export the data to ASCII-file.

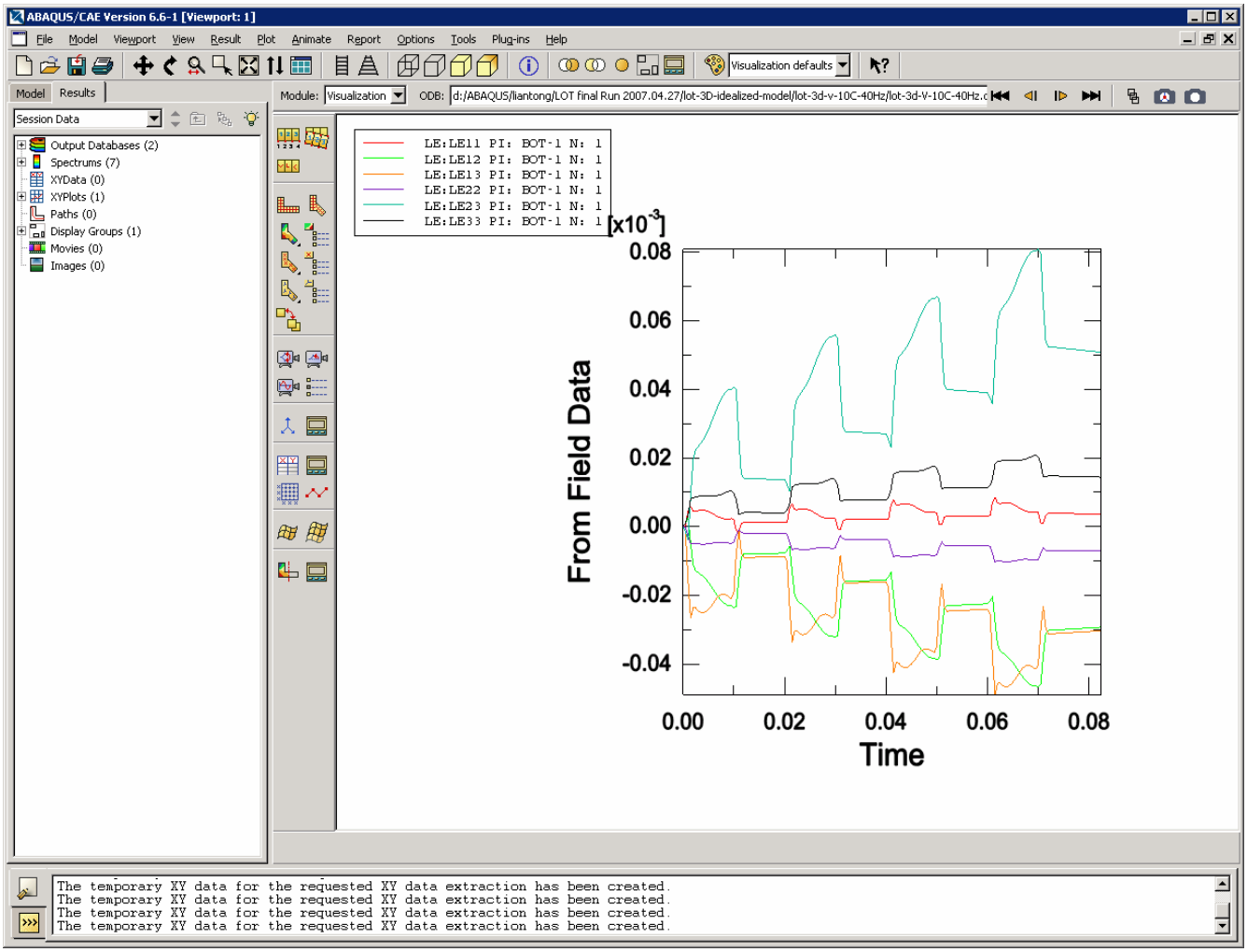


Figure 4.27 Route to ASCII-file export, opening *Report XY data* window.

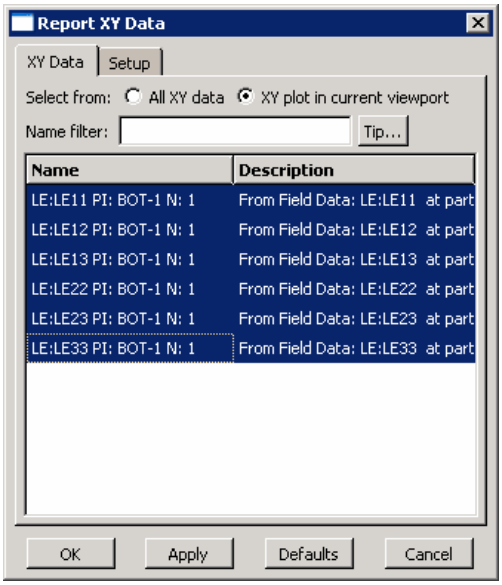


Figure 4.28 Route to ASCII-file export, the *Report XY data* window.

The process described above should result in two ASCII files that can be brought into Excel for further processing. The aim is to determine the dissipated energy as a result of the following strain-stress couples.

LE11 vs S11: ***Strain 11 and stress 11 components***
LE22 vs S22: ***Strain 22 and stress 22 components***
LE33 vs S33: ***Strain 33 and stress 33 components***
LE12 vs S12: ***Strain 12 and stress 12 components***
LE13 vs S13: ***Strain 13 and stress 13 components***
LE23 vs S23: ***Strain 23 and stress 23 components***

4.4 2D photo model

4.4.1 Adhesive zone damage

The process of exporting ABAQUS results of 2D photo models into ASCII-files is similar to the process of exporting results of idealised models. For this reasons reference is made paragraph 4.2.1. The main difference is that the 2D photo meshes may have various locations of interest. Because of this the nodes for which data needs to be exported are selected manually.
In this section the process of exporting data from ABAQUS to ASCII-file is briefly discussed. Emphasis is on how to select nodes of interest.

Fist open de relevant output data base file (odb) in ABAQUS visualization mode. Then proceed to the ***XY Data from ODB field Output*** window and select Stress components ***S22*** and ***S12*** for unique nodal positions, see figure 4.29.

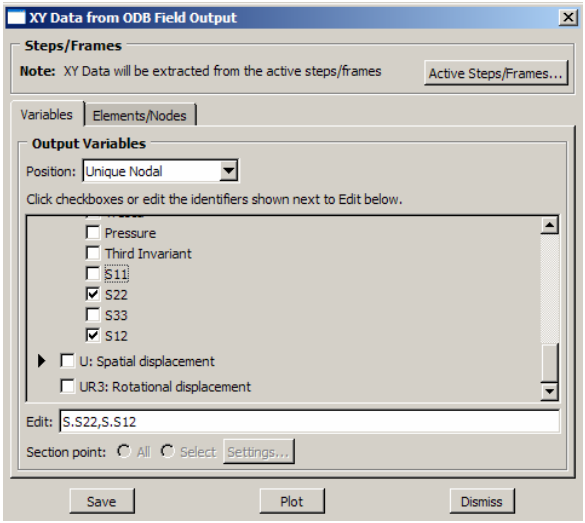


Figure 4.29 Route to ASCII-file export, the *XY data form field output* window.

Hereafter open the ***Elements/Nodes*** tab and select ***Pick from viewport*** as the method of selection. Then click ***Edit Selection***, figure 4.30. Move the ***XY Data from ODB field Output*** window so that the viewport is fully visible, figure 4.31.

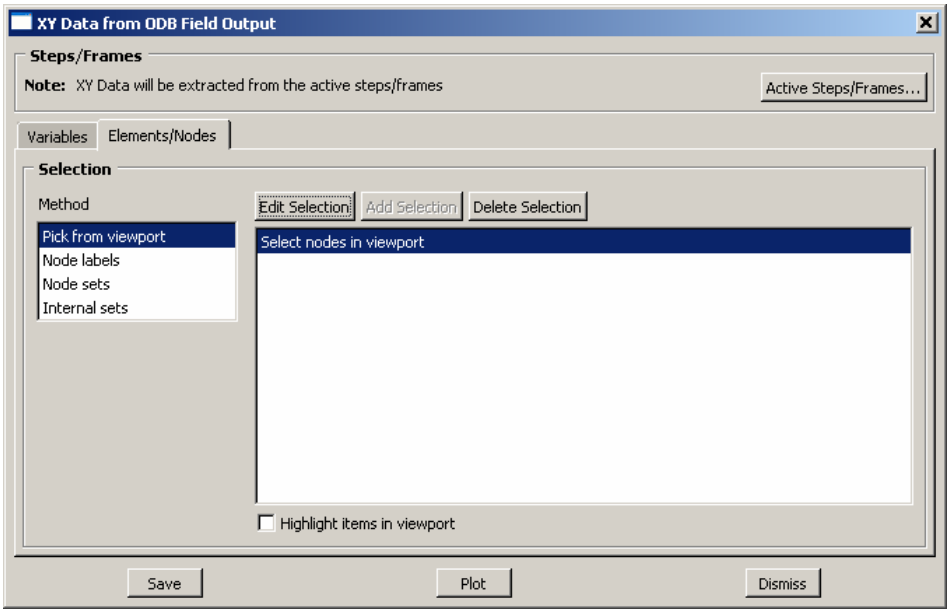


Figure 4.30 Route to ASCII-file export, select *Pick from viewport* as selection method.

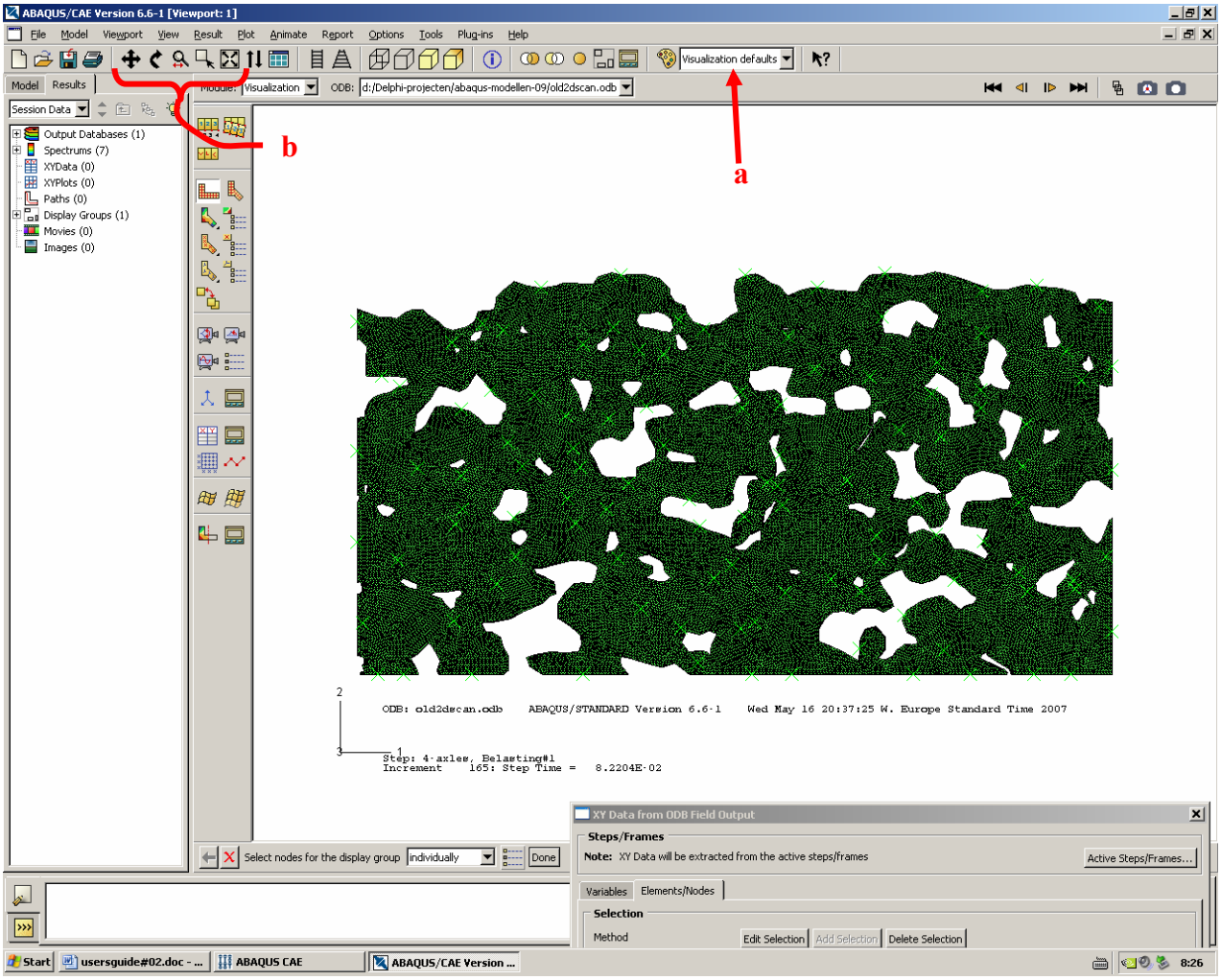


Figure 4.31 Route to ASCII-file export. A: select *Materials*. B: use zoom options to zoom in to the central surface area of the model.

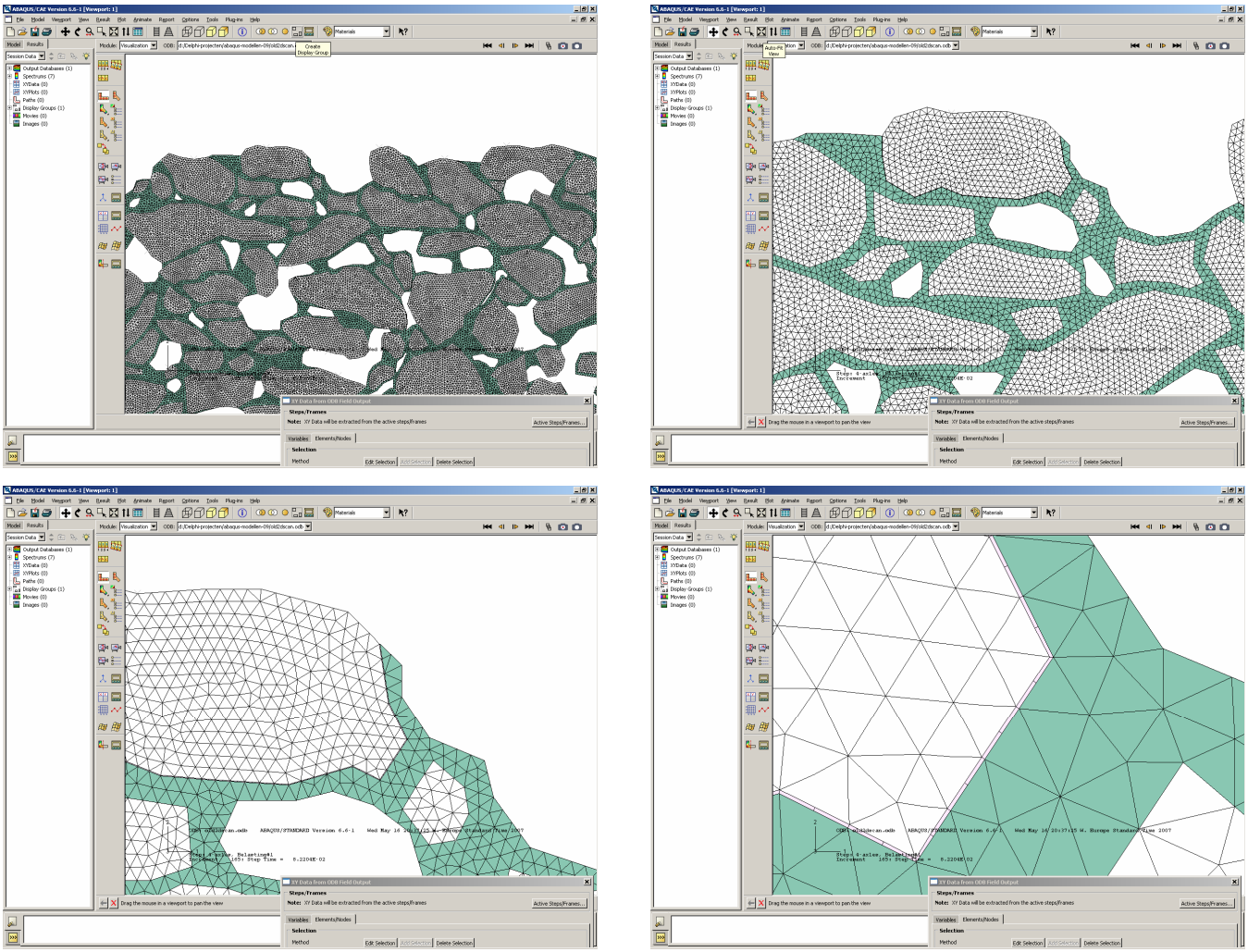


Figure 4.32 Various steps in the zoom process.

When the area of interest is displayed with ample detail, i.e. the adhesive zone can be clearly identified, select the node of interest. Now select ***Highlight items in viewport*** in the ***XY Data from ODB field Output*** window and check the location of the node of interest. The best results are obtained when the selected node is sitting on the stone surface, see Figure 4.33. After selection of the node of interest first click ***Done*** in the viewport and then click ***Plot*** in the ***XY Data from ODB field Output*** window.

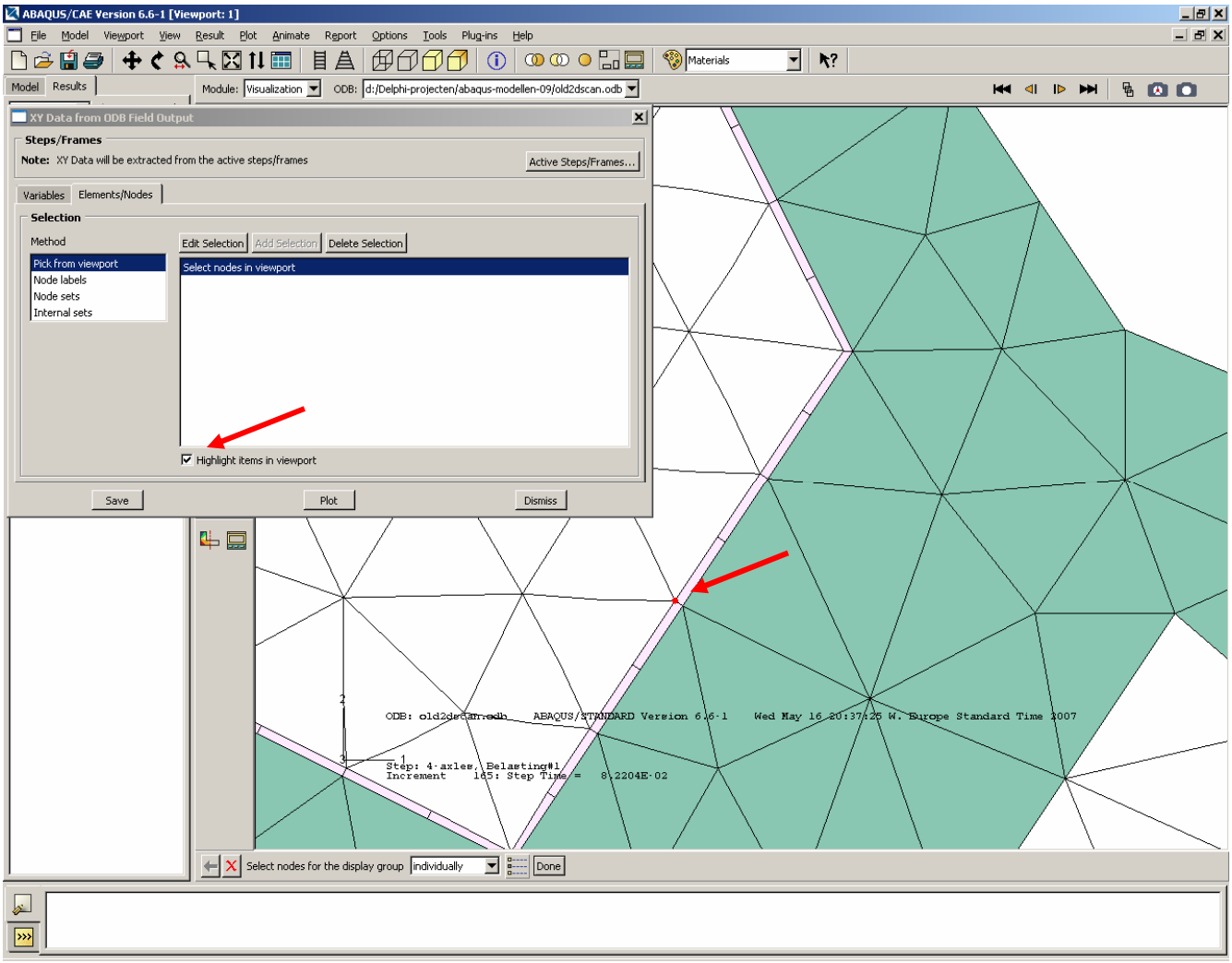


Figure 4.33 Selection of node of interest and indication of the *Highlight items in viewport* option.

Finally use the ***Report*** pull down menu to access the ***Report XY data*** window and export the plotted data to ASCII-file, see figure 4.34.

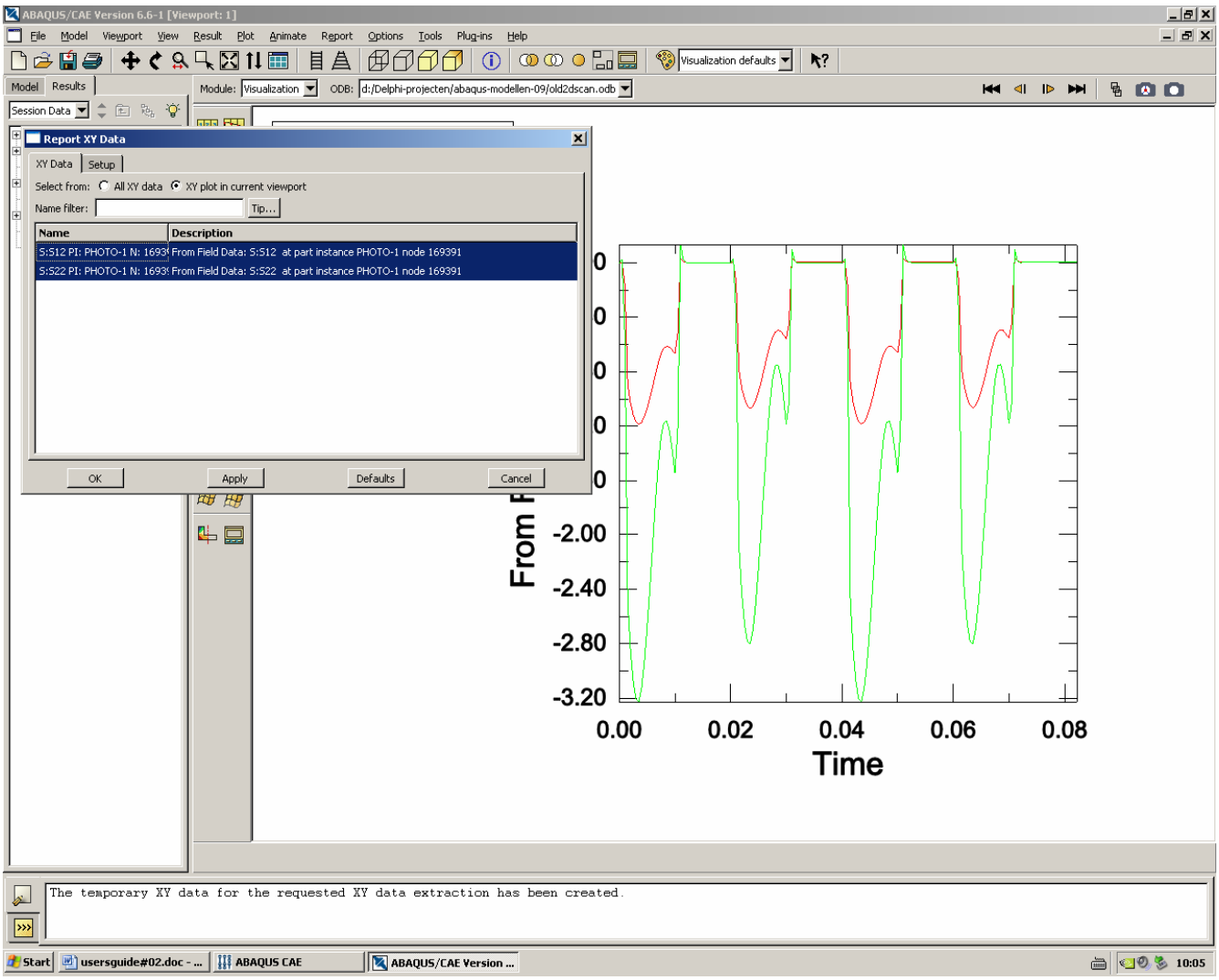


Figure 4.34 Route to export adhesive zone data to ASCII-file, the final step: *Report XY data* window.

After closure of the ***Report XY Data*** window, by clicking ***OK***, an ABAQUS report file (*.rpt) is generated in the work directory. The report file contains the desired computational data, which can be imported into Excel for further data analysis. The stress components for interface zone are as follows:

S22 = Direct through-thickness stress
S12 = Transverse shear stress

4.4.2 Mortar damage

The process of exporting ABAQUS results of 2D photo models into ASCII-files is similar to the process of exporting results of 2D idealised models. For this reasons reference is made paragraph 4.2.2. The main difference is that the 2D photo meshes may have various locations of interest. Because of this the nodes for which data needs to be exported are selected manually.
In this section the process of exporting data from ABAQUS to ASCII-file is briefly discussed. Emphasis is on how to select nodes of interest.

Fist open de relevant output data base file (odb) in ABAQUS visualization mode. Then proceed to the ***XY Data from ODB field Output*** window and select Stress components ***S11***, ***S22*** and ***S12*** for ***Unique nodal*** positions. Also select Logarithmic strain components ***LE11***, ***LE22*** and ***LE12***, Figure 4.12 and 4.13. Then open the ***Elements/Nodes*** tab and select a node in the adhesive bridge of interest, see figure 4.35. After selection of the node of interest first click ***Done*** in the viewport and then click ***Plot*** in the ***XY Data from ODB field Output*** window.

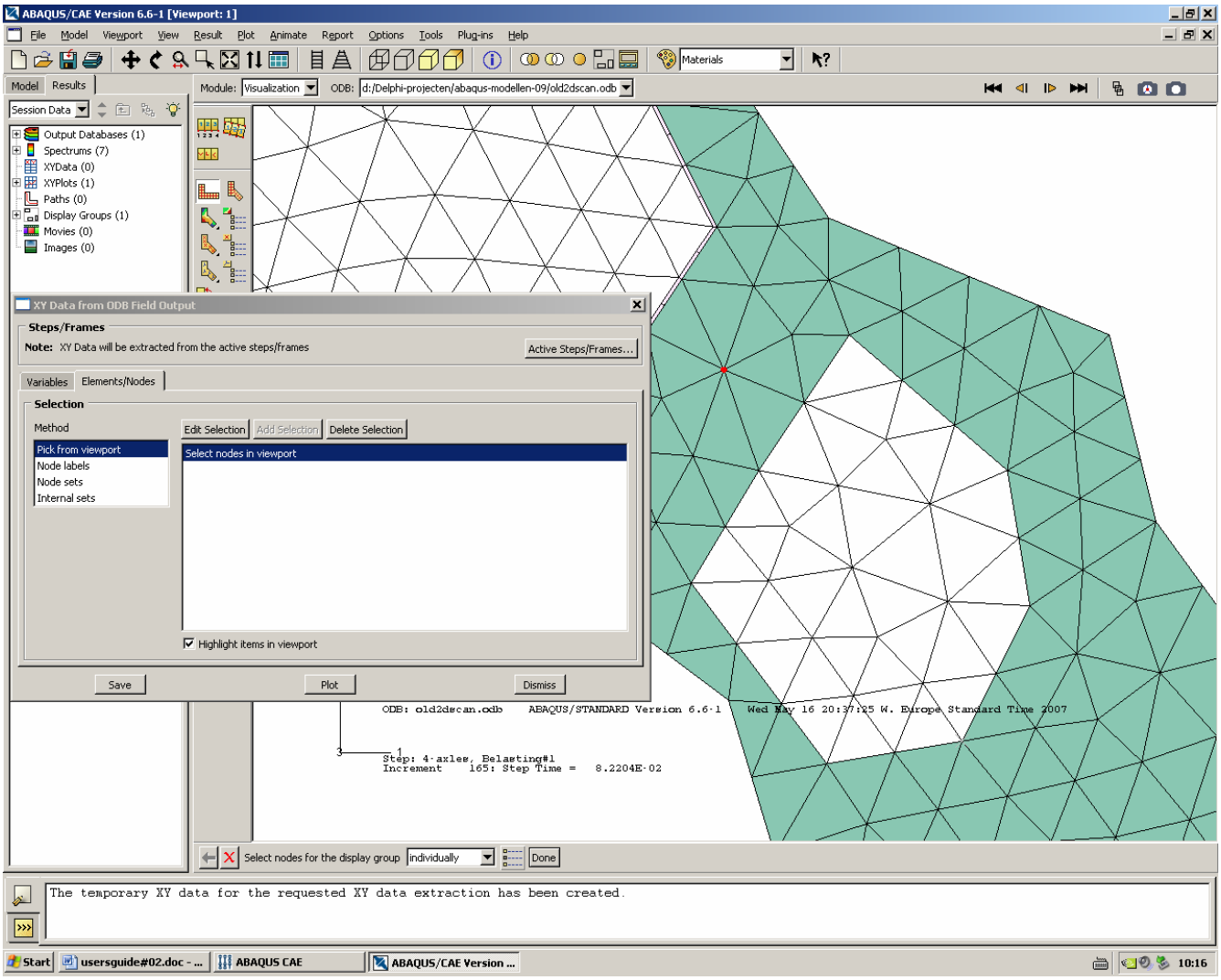


Figure 4.35 Route to export mortar bridge data to ASCII-file. Selection of the node of interest.

Finally use the ***Report*** pull down menu to access the ***Report XY data*** window and export the plotted data to ASCII-file, see figure 4.36.

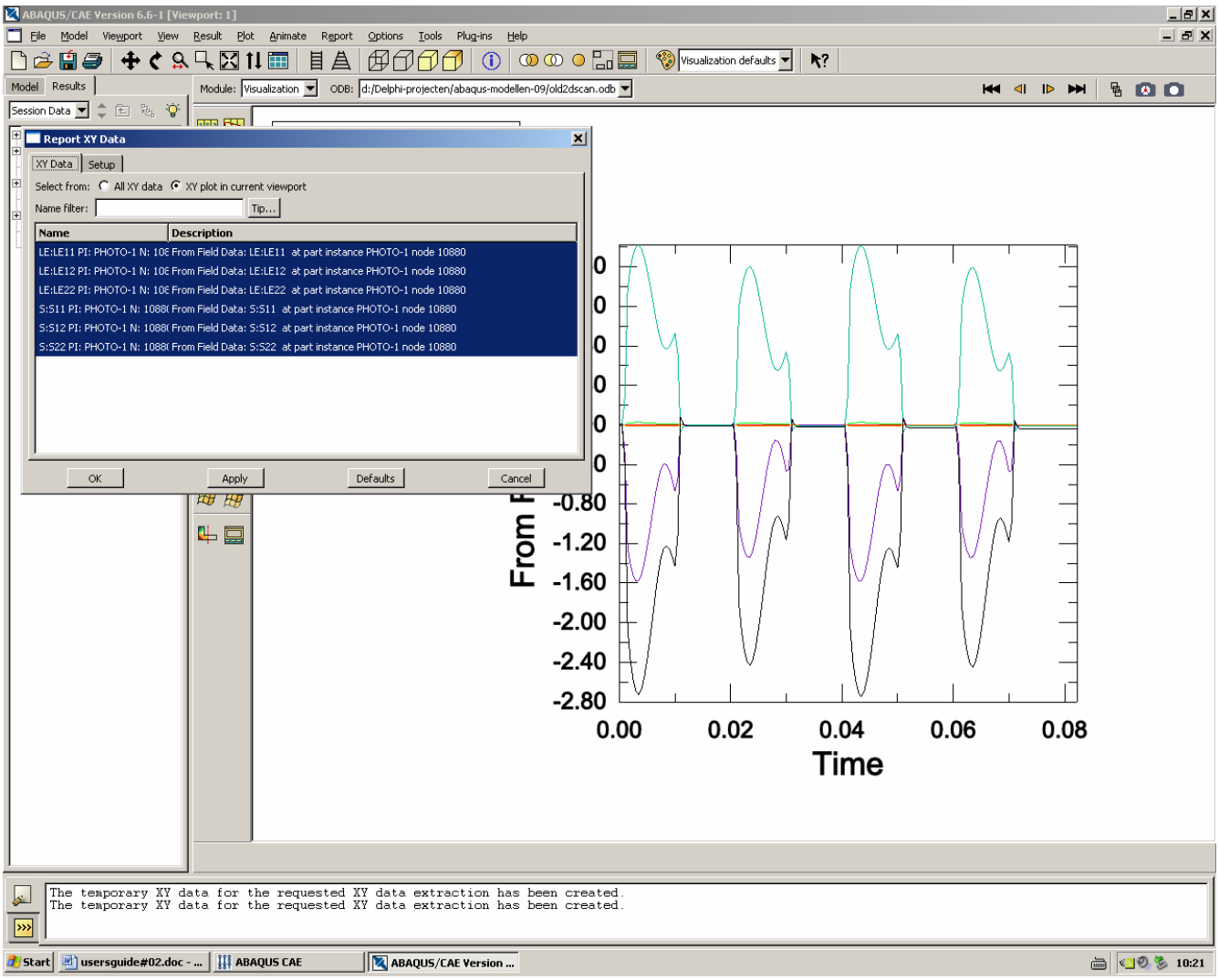


Figure 4.36 Route to export adhesive zone data to ASCII-file, the final step: *Report XY data* window.

After closure of the ***Report XY Data*** window, by clicking ***OK***, an ABAQUS report file (*.rpt) is generated in the selected directory. The report file contains the selected computational data, which can be imported into Excel for further data analysis. For determination of the dissipated energy the following signals are considered.

LE11 vs S11: ***Strain 11 and stress 11 components***
LE12 vs S12: ***Strain 12 and stress 12 components***
LE22 vs S22: ***Strain 22 and stress 22 components***

LITERATURE

1 M. Huurman, **Lifetime Optimisation Tool, LOT, Main Report**, Delft University of Technology, December 2007.

2 M. Huurman, M.F. Woldekidan, **LOT, Mortar response; measurements, test interpretation and determination of model parameters**, Delft University of Technology, December 2007.

3 M. Huurman, L.T. Mo, **Fatigue in mortar and adhesive zones; measurements, test interpretation and determination of model parameters**, Delft University of Technology, December 2007.

4 R.N. Khedoe, J. Moraal, **Lifetime Optimisation Tool, Sample preparation and laboratory testing for the LOT research program**, Delft University of Technology, December 2007.

5 ABAQUS, **ABAQUS User's Manual**, Version 6.6,